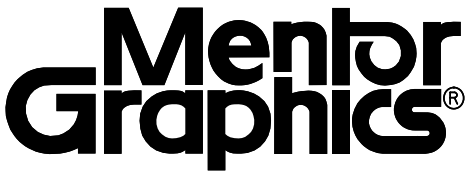
Board Station for New Users Training Series

Module 6: Routing Traces on a Circuit Board

Software Version 8.5_2



Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

TABLE OF CONTENTS

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

TABLE OF CONTENTS [Continued]

LIST OF FIGURES

LIST OF FIGURES [Continued]

LIST OF FIGURES [Continued]

LIST OF TABLES

About This Training

Welcome to the *Board Station for New Users Training Series*. For information on the tools you learn to use in this training series, see the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Workbook Organization

For an overview of the organization and content of all the modules of the *Board Station for New Users Training Series*, refer to section "Workshop Overview" in *Module 1: Introduction to Board Station.*

Related Documentation

For a complete listing of the manuals that make up the PCB documentation set, refer to section "Guide to the Documentation" in the *PCB Products Overview Manual*. The *PCB Products Overview Manual* describes how each manual can help you in the design process. You can find a listing of all Mentor Graphics manuals in the *Mentor Graphics Technical Publications Overview Manual*. Both of these manuals are available in INFORM.

Documentation Conventions

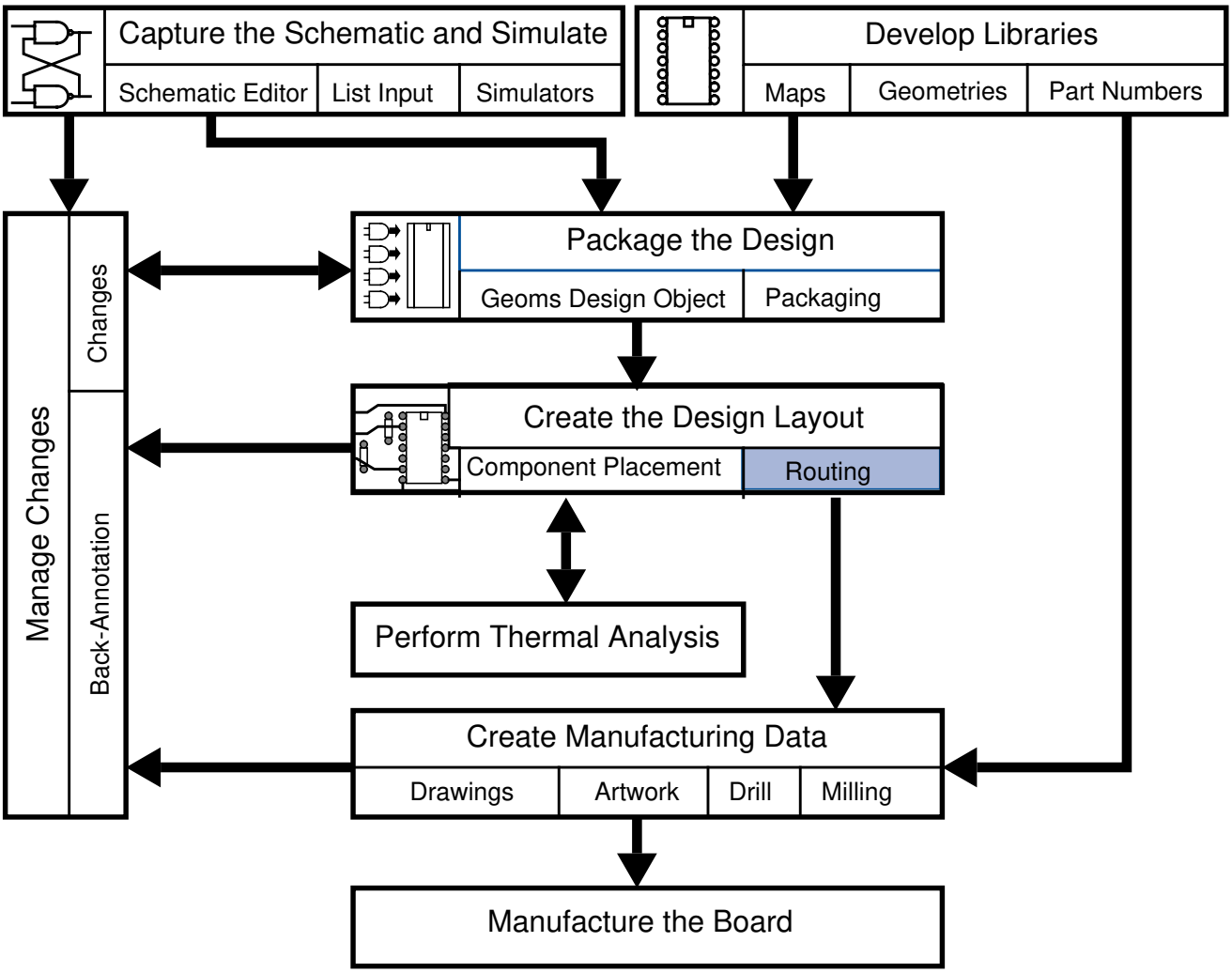
For an explanation of the documentation conventions used in this workbook, refer to the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Installation Procedure

For complete instructions on installing the data for this module, refer to "Installation Procedure" in the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Lesson 1
Design Rules and Interactive Routing

The Routing Traces on a Circuit Board module is the second part of the discussion on the circuit board design process. This part of the design process includes interactive and automatic routing, routing traces with the Dynamic Editor, and creating area fills. Lesson 1 discusses how to set up for routing, as well as how to interactively route your design.



Objectives

This lesson of the routing module covers design rules and interactive routing. Your design rules are saved as a part of your design data in a file called a *tech* design object.

In this section, you examine the types of design rules and the methods for creating and modifying the design rules. Attributes and properties also control the behavior of your design during the routing process. This section explains various attributes and properties that are used to set up for the routing process. You also examine the techniques for interactive routing.

After completing this module, you are able to do the following:

- Describe the *tech* design object.
- Explain attributes and properties used for routing.
- Explain the setup parameters for routing.
- Explain interactive routing.

Design Rules

The *tech* design object stores the design rules. To review all of the existing design rules for your design, choose the **View > Design Data** menu item and select the *tech* design object from the resulting list of design objects. Press the OK button.

The categories of design rules are:

- **Physical layer rules**—define the stacking order of the physical board layer.
- **Pin rules**—identify the layers spanned by the padstack, designate layers on which to prevent connections, and specify whether the pin rule applies to top or bottom placement of a component.
- **Via rules**—identify the layers spanned by the via and designate the layers on which the via can connect directly to a single-layer pin padstack.
- **Layer rules**—allow or suppress routing on a layer and set the preferred routing direction on a layer.
- **Net rules**—assign design rules like trace width, pin, via, and trace clearances that apply to the nets in a design.
- **Net rules for layers**—assign design rules that apply to nets on a specific layer.

Physical Layers

By default, the system creates a physical layer structure based on the signal and power layers defined for your board geometry. This structure is automatically saved in the *tech* design object during any normal exit from LAYOUT and restored whenever LAYOUT is invoked.

Use the Setup Physical Layers dialog box to add, delete, or change physical layers in your design.

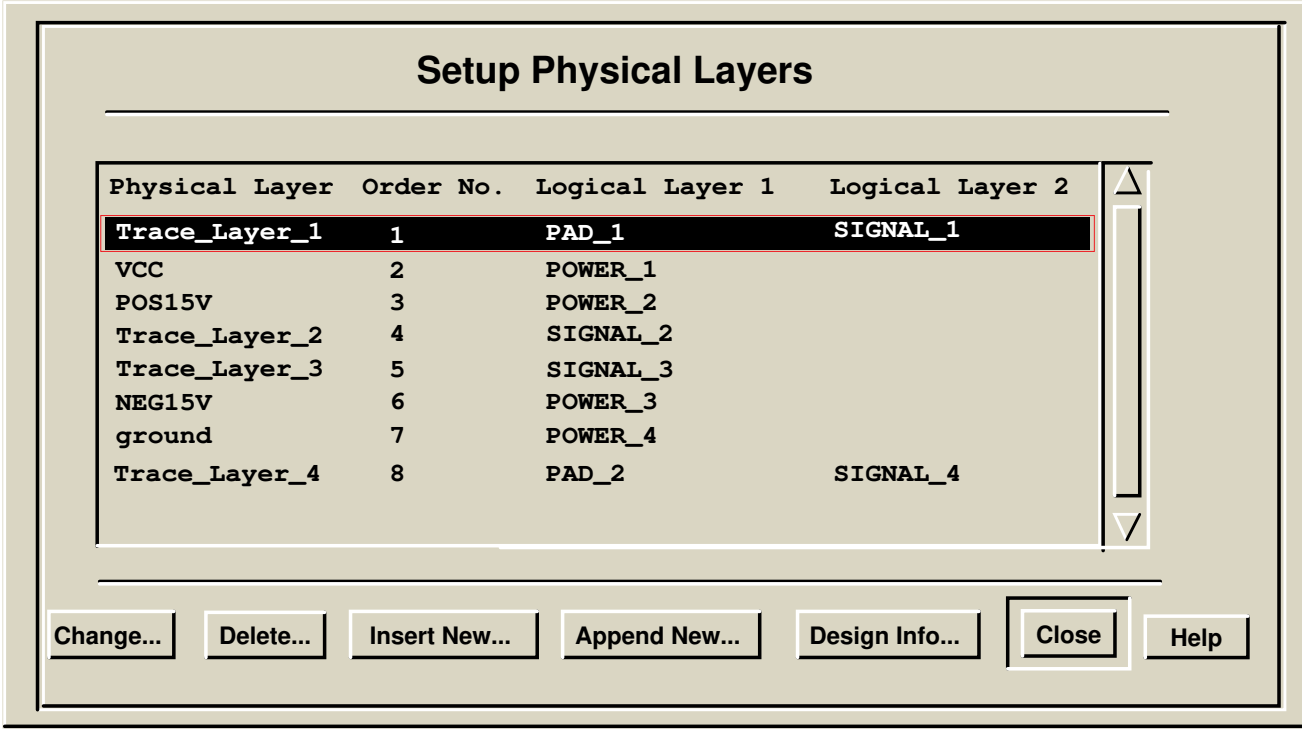


Figure 1-1. Setup Physical Layers Dialog Box

The order number indicates where a specific layer exists in the stack of layers that make up the physical board. For example, layer Trace_Layer_1 is the top layer in the board. The Board Station logical layers used to create physical layer Trace_Layer_1 are logical layers Pad_1 and Signal_1.

Default Physical Layers

Physical layers are numbered from 1 (for the top surface layer) to *n* (for the stacking order number of the bottom surface layer). The default physical layer names use the stacking order number, such as Physical_1 for the top physical layer, Physical_2 for the second physical layer in the stacking order, and ending with Physical_*n* for the bottom physical layer.

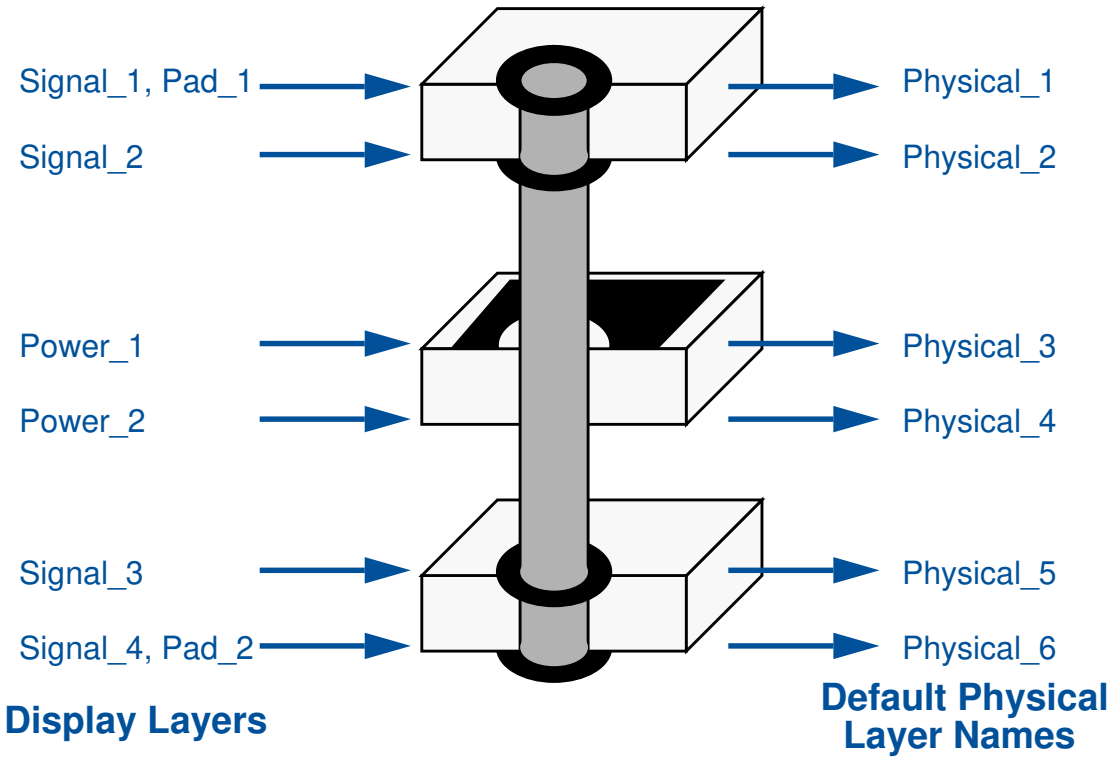


Figure 1-2. Mapping Logical Layers to Physical Layers

With the exception of the top and bottom (surface) physical layers, only one logical layer is associated with each physical layer. The default scheme assigns the logical signal layer and power layer names to physical layers. For example, Signal_2 is assigned to Physical_2. For the top and bottom layers, the system also assigns the Pad_1 and Pad_2 logical layers.

When assigning power layers to physical layers be sure to assign the power layers sequentially.

Add/Delete/Rename Physical Layers

You can add new physical layers to your design by inserting them into the current list at any level, or by appending them to the list. The logical layers you associate with the physical layers must be included in the *layers* design object for your design. You can use this capability to add new routing layers.

You receive warning messages on invocation if added routing layers exceed the number specified in the Board_routing_layers attribute of your board geometry. However, you do not need to change this attribute in order to use the new routing layers.

When you delete a physical layer, any routing that was on the deleted layer is also deleted and replaced with guides. A guide is a line that the system provides to indicate connectivity between two pins. When you create a trace between those two pins, the guide is removed. Vias on deleted layers are replaced with z-axis guides. Buried vias spanning the deleted layer(s) are not deleted if a larger usage rule exists and can be used. If this is not possible, then the buried vias are deleted and replaced with z-axis guides.

If you delete a power layer, you receive a warning message that the number of power nets exceeds the number of power layers. If you delete a surface layer, the associated pad layer is reassigned to the next available logical signal layer. If no signal layer is available, the pad layer remains unassigned.

You can change the name of any physical layer, as long as you do not duplicate physical layer names already in your design.

Rules for Pins and Vias

Usage rules for pins and vias work in conjunction with the physical layer structure to define where and how the router can connect to padstacks and how vias are used.

The system defines default rules for the pin and via padstacks in your design, based on pin or via padstack type. Depending on the type of padstack, you can change and add to these rules to create your own user-defined usage rules.

To determine the types of via padstacks in your design, choose the **Report > Routing > Via Rules** menu item.

To define via rules, select the **Setup Routing > Physical Layers > Physical Layers...** menu item.

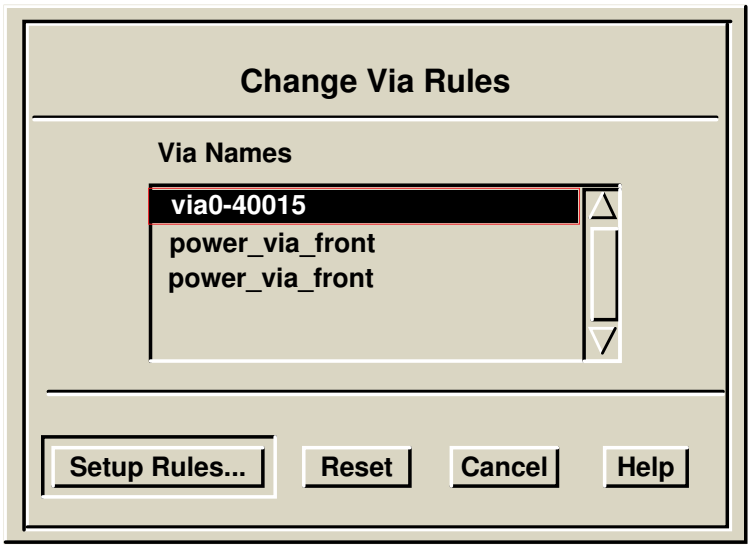


Figure 1-3. Change Via Rules Dialog Box

You need to know what type each padstack is before you can edit a specific padstack.

All usage rules are stored in the *tech* design object.

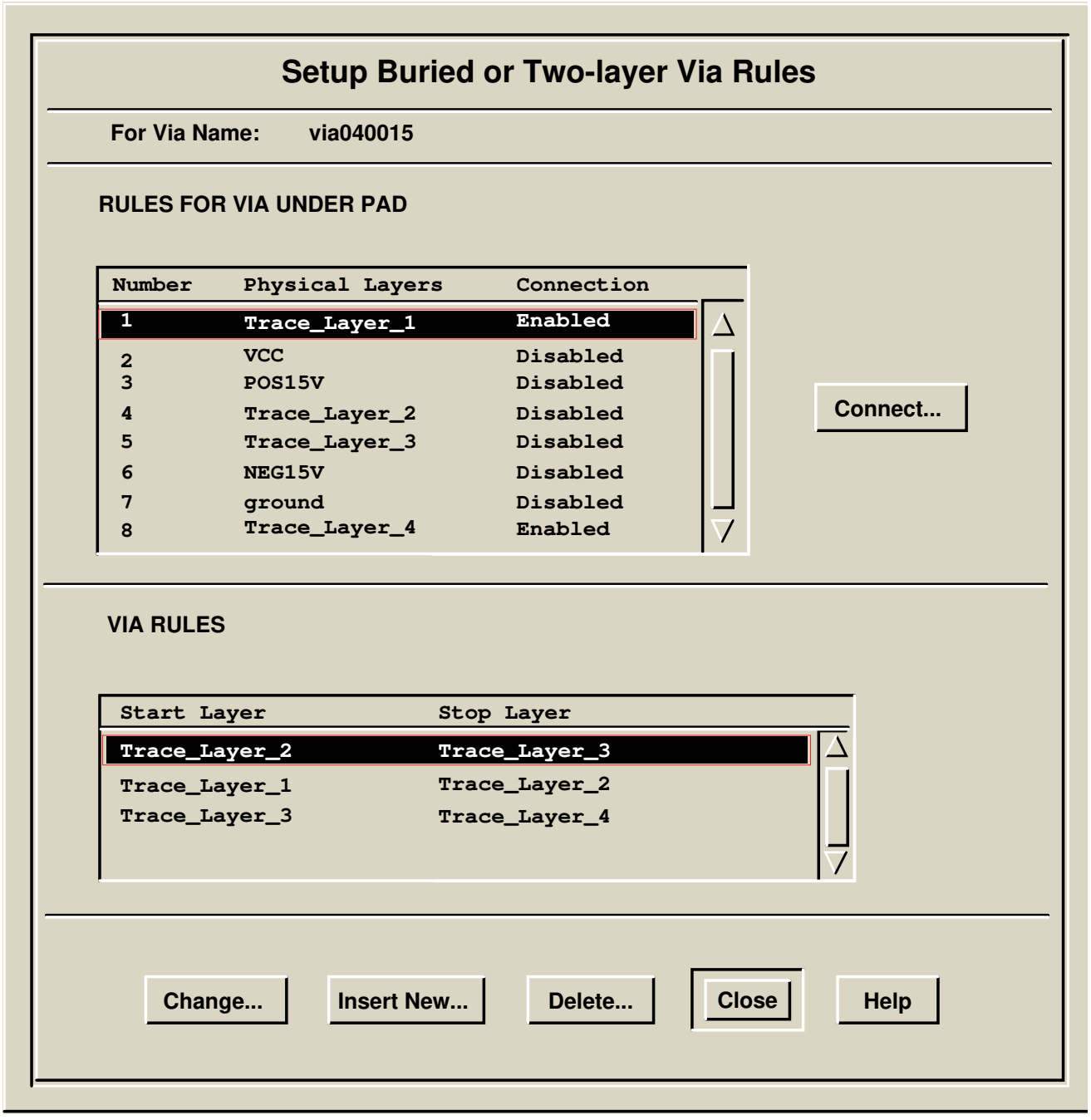
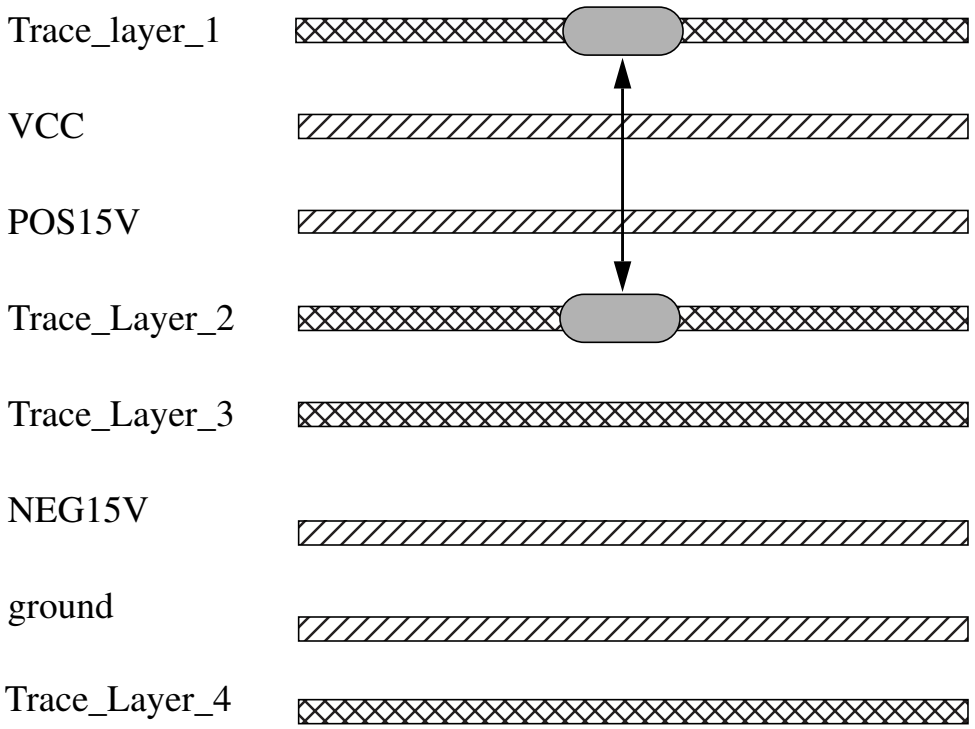


Figure 1-4. Setup Via Rules Dialog Box

To determine the types of pin padstacks in your design, choose the **Report > Routing > Pin Rules** menu item.

Understanding Blind/Buried Pins



Pin rule connects blind pin fromTrace_Layer_1 to Trace_Layer_2.

Figure 1-5. Blind Pin Rule

Blind or buried pins are pins that attach to multiple layers of the board, but do not go completely through all layers like a through pin. Blind pins and buried pins are the same type pin. The difference is only the layers on which the pin appears in a particular design. A pin is considered a blind pin if it begins or ends on an outer layer. The same pin is considered a buried pin if it both begins and ends on internal layers.

Blind/buried pins are made up of two parts: the graphics and the pin rules. The graphics for a blind/buried pin are usually created on the generic Signal layer, but can be created on any signal layer. Pins rules then control the layers on which the pin appears graphically as well as the layers to which the pin connects. A blind/buried pin can have only one set of pin rules: one rule if the pin is placed on the top of the board and one rule if the pin is placed on the bottom of the board.

Default Pin Rules

By default, the system defines rules for all pin padstacks based on the pin padstack type, which is determined by the attributes associated with the padstack.

Table 1-1. Pin Rule Defaults by Pin Type

Default Pin Rules		
Type	**Attribute**	**Default Rules**
Through Pin	Terminal_thruhole_definition	Top-to-Bottom
Surface Pin	Terminal_surface_definition	Top-to-Top Bottom-to-Bottom
Blind Pin	Terminal_blind_definition	Top-to-Top Bottom-to-Bottom

If you define no pin rules, the default rules apply to all pin padstacks. You cannot change the pin rules for surface pin padstacks. However, you can redefine rules for blind pin padstacks and extend rules for through pins to disallow connections on specific layers.

Defining Pin Rules

You can edit the pin rules for any blind pin padstack and extend rules for through pins shown in the popup pin rules report. To get a popup pin rules report, choose the **Report > Routing > Pin Rules** menu item. A blind pin refers to any pin padstack with the Terminal_blind_definition attribute. A through pin refers to any pin padstack with the Terminal_thruhole_definition attribute. Only pin padstacks with these attributes are listed in the report.

Editing pin rules is a three- or four-step process in which you progress through a series of dialog boxes. To edit pin rules:

1. Choose the **Setup Routing > Physical Layers > Pin Rules** menu item.

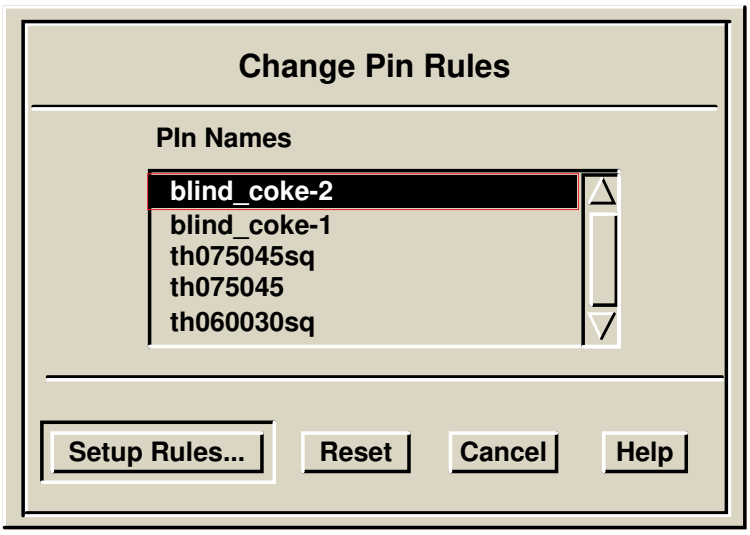


Figure 1-6. Change Pin Rules Dialog Box

The Change Pin Rules dialog box displays. It lists all of the blind and through pin padstacks found in the *geoms* design object.

2. Select one pin from the list in the Change Pin Rules dialog box, and then press the **Setup Rules...** button at the bottom of the dialog box.

The Setup Pin Rules dialog box appears, displaying the current rules for the previously selected pin padstack.

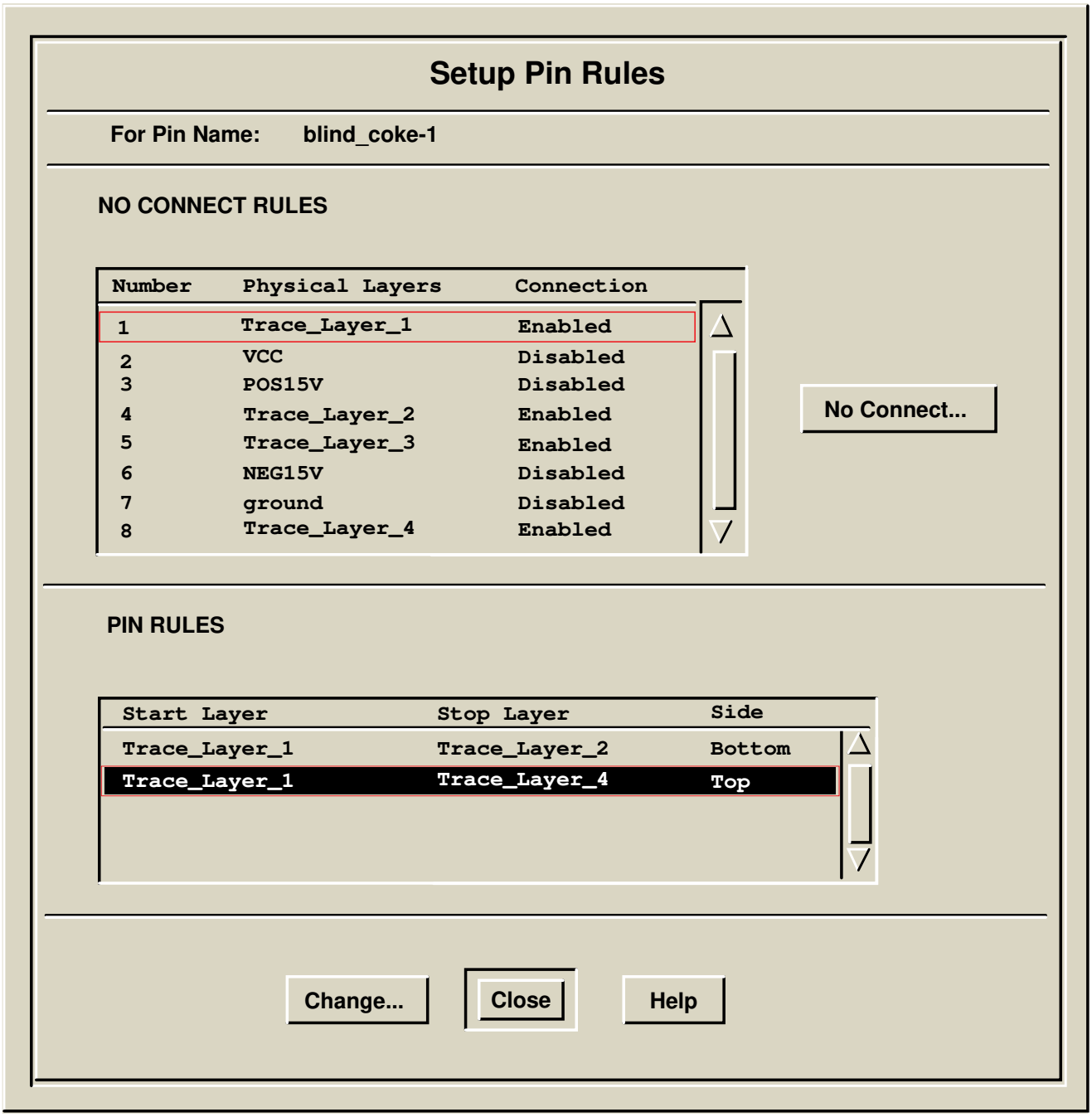


Figure 1-7. Pin Rule Definition

The contents of this dialog box depends on the type of pin padstack you selected from the Change Pin Rules dialog box. If you select a through pin, you can only extend rules to disallow connections on specific layers. If you select a blind pin, you may redefine all the rules.

3. Select one rule to change, or choose to specify noconnect rules for the pin.

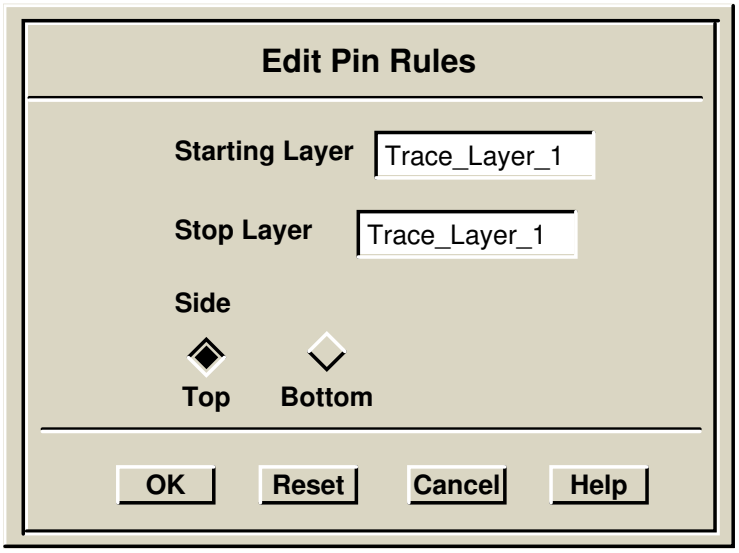


Figure 1-8. Pin Rule Editing

Use the Edit Pin Rules and No Connect Pin Rules dialog boxes to alter all the usage rules shown in the Setup Pin Rules dialog box.

For more information on editing via padstacks, refer to the information for defining pin rules in *Using PCB Design Tools.*

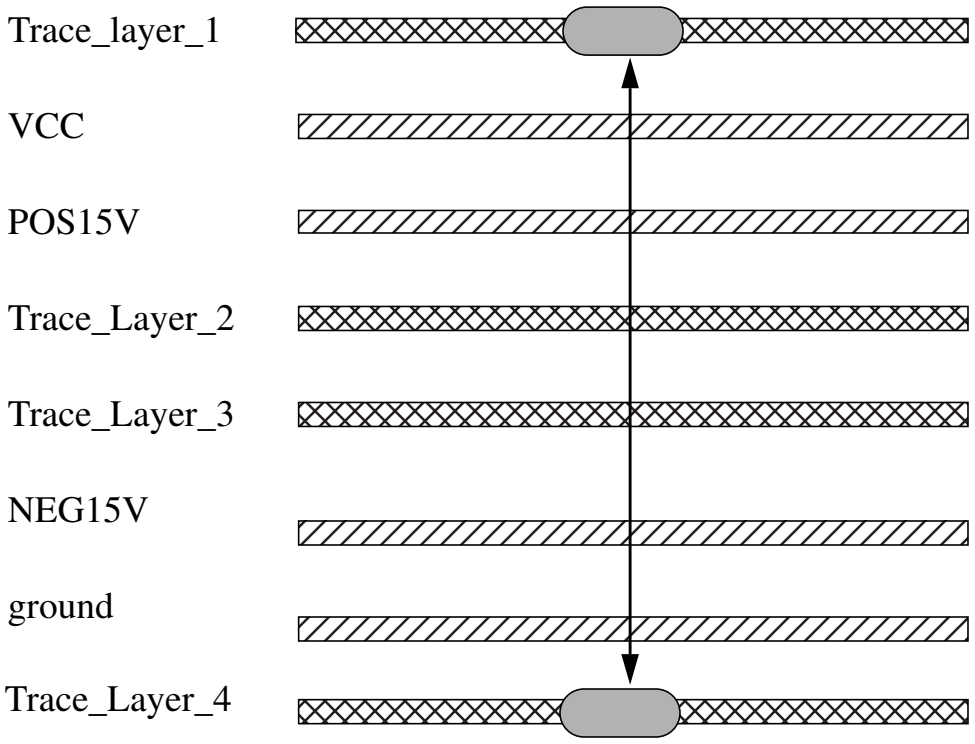
Multiple Pin Rules for a Single Pin

It is possible to define rules that allow a blind pin padstack to appear as a surface pin, as a blind pin, or as an internal pin (appearing only on internal layers). When editing pin rules, the following requirements must be met:

- Each blind pin padstack geometry must have exactly two pin rules.
- Each rule must start and end on a valid physical layer.

Use the Edit Pin Rules dialog box to modify the selected usage rules for the pin.

Understanding Noconnect for Pins



Connection is disabled for layers Trace_Layer_2 and Trace_Layer3.

Figure 1-9. Disabling Connections to Layers

Now, you change the pin rule to connect the blind/buried pin between layers Trace_Layer_1 and Trace_Layer_4. But, you do not want connections on layers Trace_Layer_2 or Trace_Layer_3. Selecting the **No Connect...** button in the Setup Pin Rules dialog box displays another dialog box that allows you to disable connections to specific layers. By disabling connections to layers Trace_Layer_2 and Trace_Layer_3, the pad shape only appears on layers Trace_Layer_1 and Trace_Layer_4.

An example use of Noconnect is a blind pin for an edgeclip type connector. You want the pin to connect to the outermost top and bottom layers of the board, but not to connect to layers between the outermost top and bottom layers. Using Noconnect you can disable connections to specific layers.

No Connect

You can specify that blind pins are not allowed to connect to traces on specific layers. Press the No Connect button and choose All Layers or Specific Layers in the No Connect Pin Rules dialog box.

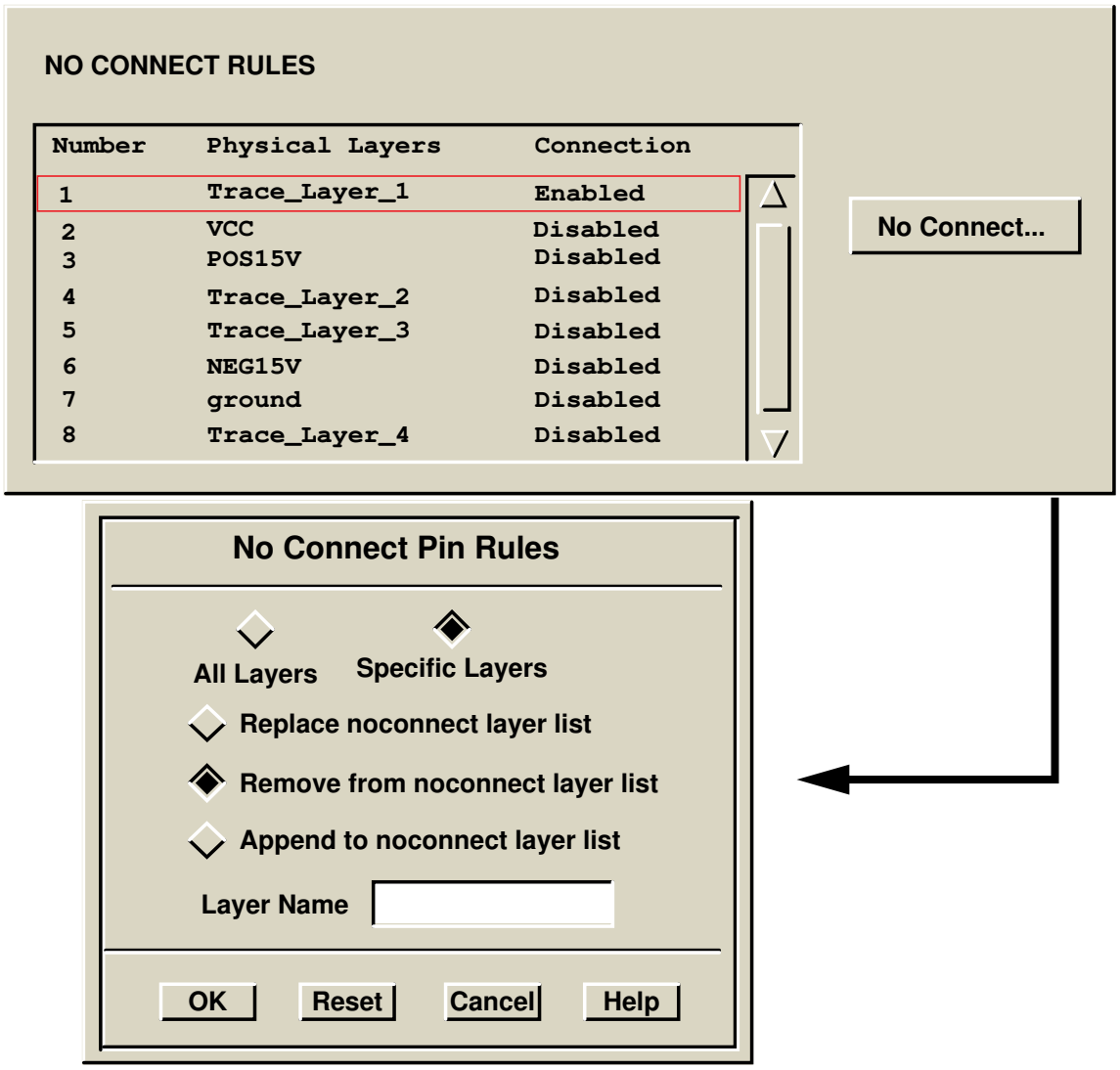


Figure 1-10. Disallowing Connections to Specific Layers

Understanding Blind/Buried Vias

Blind or buried vias are vias that attach to multiple layers of the board, but do not go completely through all layers like a through via. Blind vias and buried vias are the same type via. The difference is only the layers on which the via appears in a particular design. A via is considered a blind via if it begins or ends on an outer layer. The same via is considered a buried via if it both begins and ends on internal layers.

Blind/buried vias are made up of two parts, the graphics and the via rules. The graphics for a blind/buried via are usually created on the generic Signal layer. Via rules then control the layers on which the via appears graphically as well as the layers to which the via connects. A blind/buried via can have several via rules.

Refer to Figure 1-11. You can legally define via rules for a single via between the following layers:

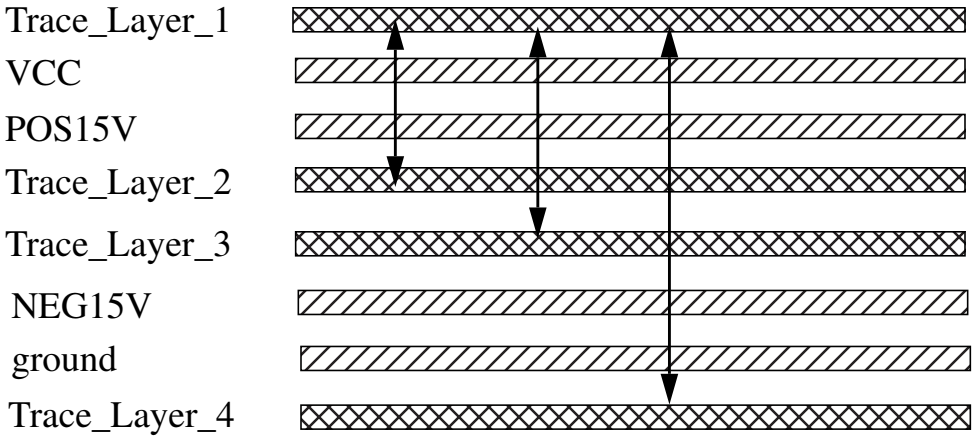
- Layers metal_1 and metal_2.
- Layers metal_1 and metal_3.
- Layers metal_1 and metal_4.

You cannot legally define one via rule that is completely within another via rule. For example:

- Layers metal_1 and metal_4.
- Layers metal_2 and metal_3.

Shapes appear on all layers covered by the via rule.

Legal via rules:



Illegal via rules:

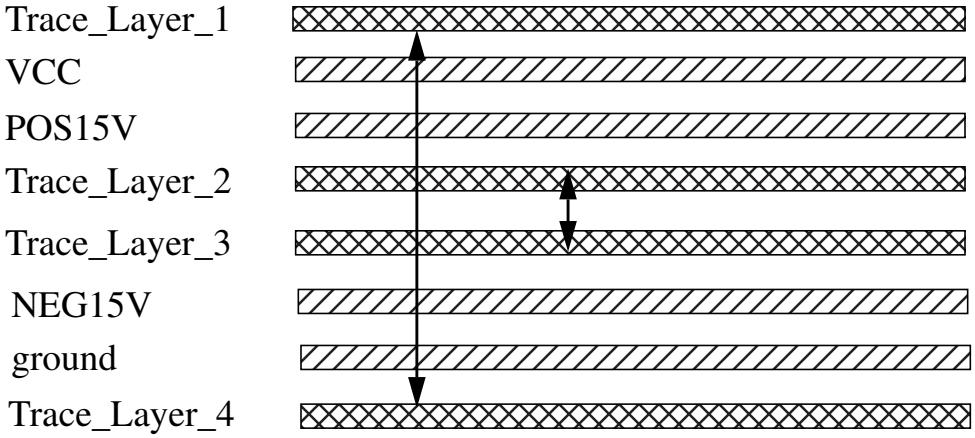


Figure 1-11. Legal and Illegal Via Rules

Understanding Two-layer Vias

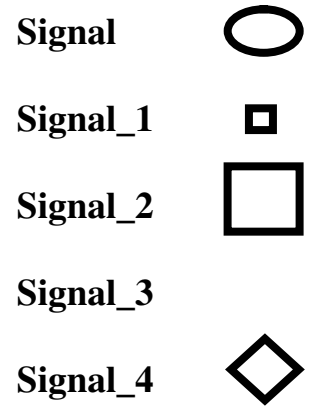


Figure 1-12. Defining Two-layer Vias

The two-layer via type connects only to the layers defined in the via rule. Shapes appear on only the two layers defined in the via rule. No shapes appear on any layer other than the two layers specified in the via rule, including layers between those specified in the rule. This prevents holes from being generated in dielectric on intervening layers.

You draw the graphics for a two-layer via on the generic Signal layer. You then define a via rule stating between which two layers the via connects.

Understanding Connect... for Vias

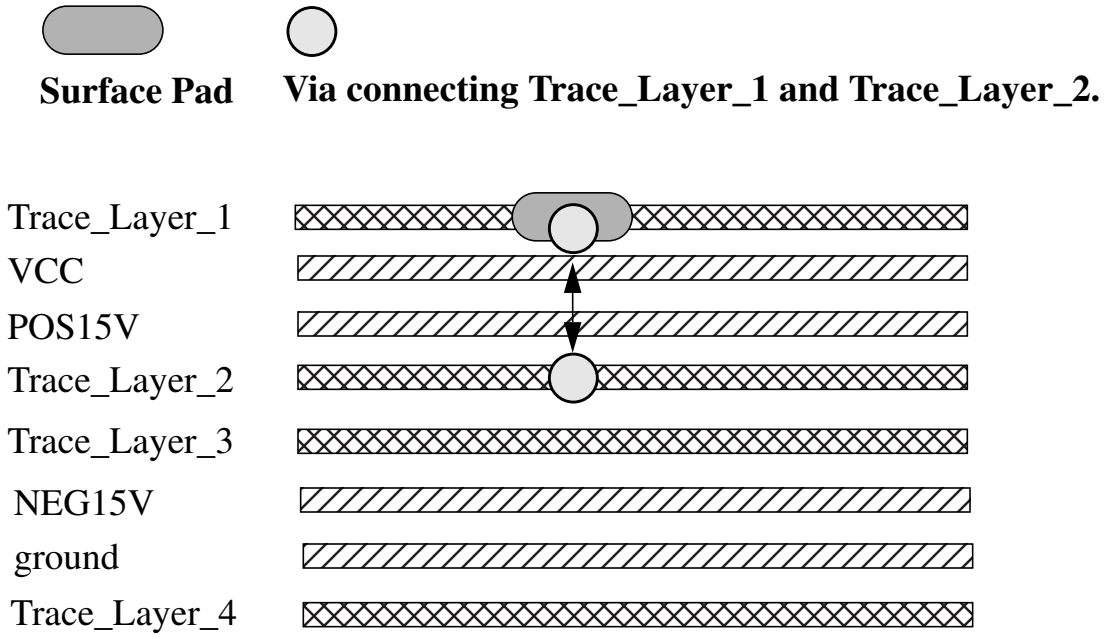


Figure 1-13. Connecting a Surface Pad to Internal Layers

You can use the **Connect...** button in the Setup Via Rules dialog box to enable a via to connect directly to a single-layer pad. In Figure 1-13, a via connects to a surface pad on layer metal_1. The via connects between layers Trace_Layer_1 and Trace_Layer_2.

Default Via Rules

By default, the system defines rules for all via padstacks, except buried and two-layer via[1] padstacks, based on the via padstack type. Padstack type is determined by the attributes associated with the padstack.

Table 1-2. Via Rule Defaults by Via Type

Type	Default Via Rules Attribute	Default Rules
Through Via	Terminal_thruvia_definition	Top-to-Bottom
Buried Via	Terminal_buried_via_definition	None
Two-layer Via	Terminal_2_layer_via_definition	None

You cannot edit the default rules assigned to a segmented via padstack.

The system does not define rules for buried or two-layer via padstacks, so if your design uses buried or two-layer vias you must define usage rules for them. You can define the rules for buried or two-layer via padstacks in either LIBRARIAN or LAYOUT. Via padstacks that do not have one or more valid rules are not available to either the automatic or interactive router.

You can further define a rule for a through via padstack to allow it under a single-layer pad.

1. Two-layer via padstacks are recognized for use only with the Hybrid option.

Defining Via Rules

You can define, alter, and delete via rules for any buried or two-layer via padstack. You can extend the default via rule for through vias to allow placement of the via under a single-layer padstack. Segmented via padstack rules cannot be altered. To get a via rules report showing all the types of vias in the currently open design, choose the **Report > Routing > Via Rules** menu item.

Buried via refers to any via padstack with the Terminal_buried_via_definition attribute, and two-layer via refers to any via padstack with the Terminal_2_layer_via_definition attribute. A through via has the Terminal_thruvia_definition attribute. Only via padstacks with these attributes are listed as buried vias, two-layer vias, and through vias in the report.

Editing via rules is a three- or four-step process in which you progress through a series of dialog boxes.

1. To edit via rules, choose the **Setup Routing > Physical Layers > Via Rules** menu item.

   The Change Via Rules dialog box displays listing all of the buried, two-layer, and through via padstacks found in the *geoms* design object.

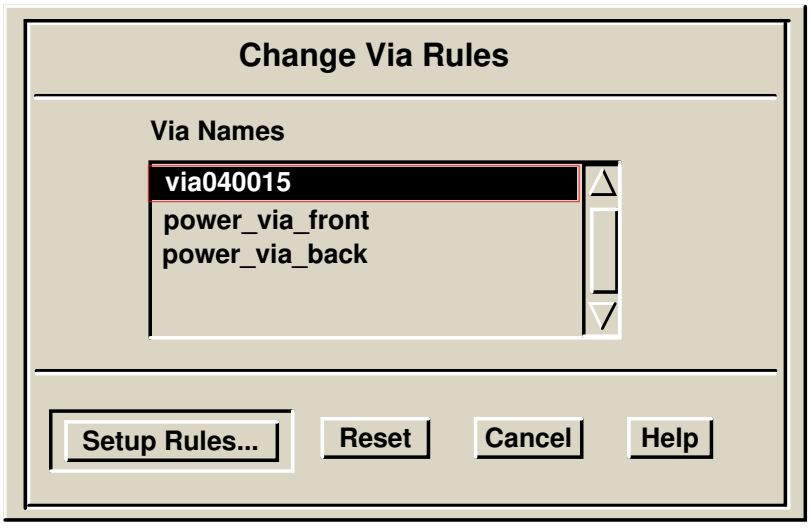


Figure 1-14. Change Via Rules Dialog Box

2. In the dialog box, select one via to edit, and then press the **Setup Rules** button.

   The Setup Via Rules dialog box displays the current rules for the previously selected via padstack. The contents of this dialog box depends on the type of via you select. If you select a buried or two-layer via, it appears as shown in the example. If you select a through via, the Via Rules portion of the dialog box does not display, and neither do the Delete or Add buttons.

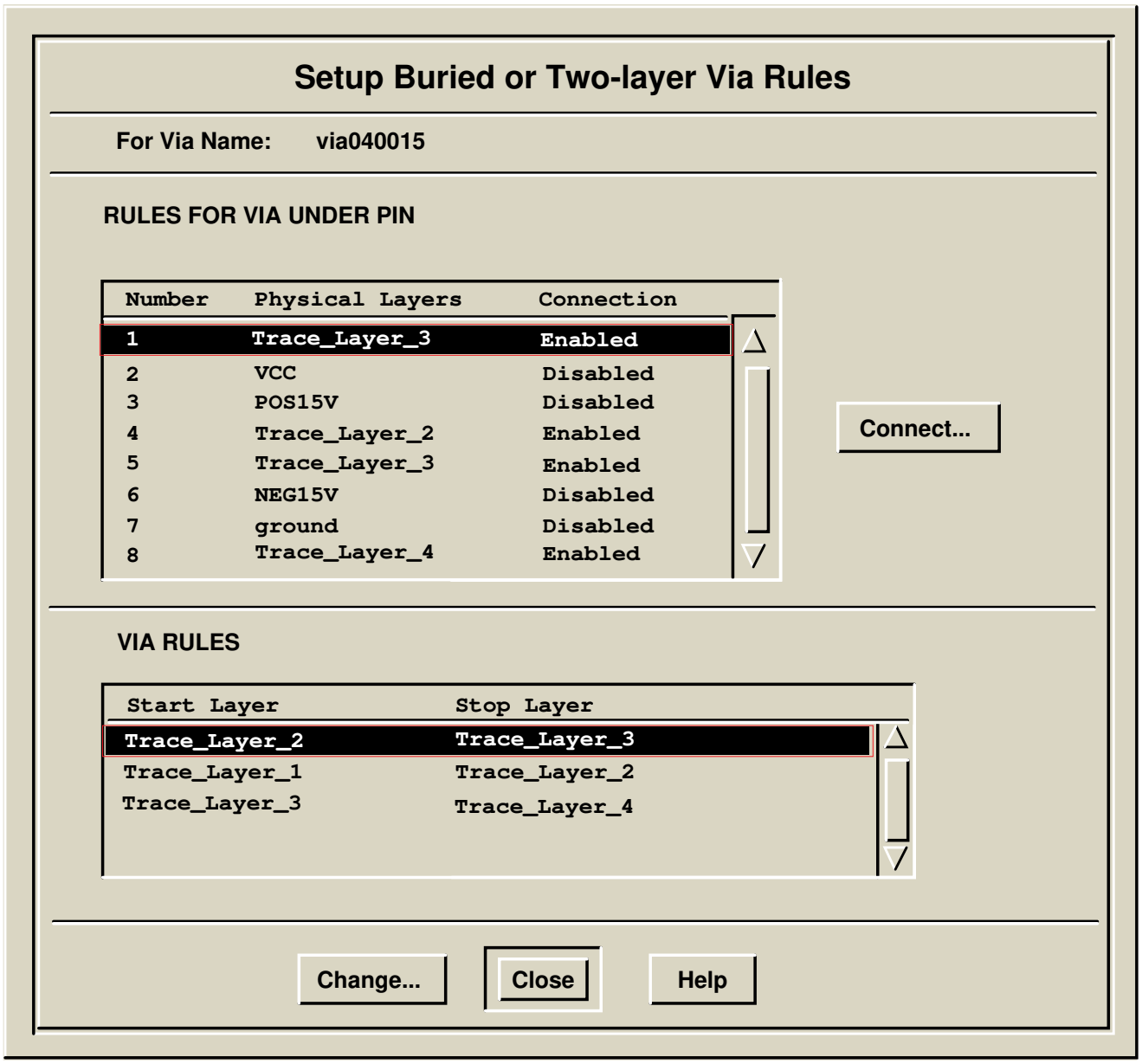


Figure 1-15. Via Rule Definition

If you select a buried or two-layer via, selection within this list indicates the rule to delete when **Delete** is selected.

For more information on editing via padstacks, refer to the information for defining via rules" in *Using PCB Design Tools.*

Multiple Via Rules for a Single Via

Via rules allow you to use a single buried via padstack geometry as a through via, as a blind via, as a buried via, or as a segmented via.

Figure 1-16. Via Rule Setup

The Add Via Rules dialog box allows you to define additional usage rules for the selected buried or two-layer via padstack.

Connect Via

The Connect Via rule allows a single-layer blind pin padstack or a surface pin padstack to route to or from a surface-mount component pin through a via placed directly beneath the padstack.

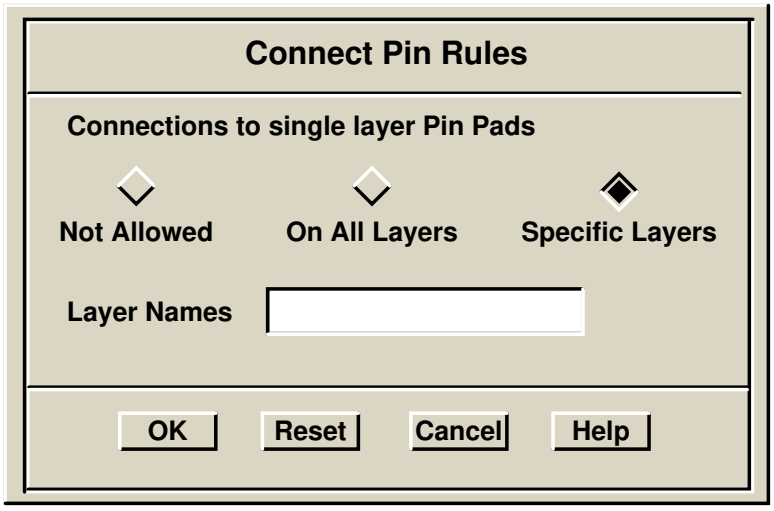


Figure 1-17. Allow Connection to a Padstack on a Specific Layer

The Connect Via Rules dialog box allows you to specify layers on which the selected via padstack can connect directly to single-layer blind pin padstacks or surface padstacks.

Rules for Layers

Use the Setup Layer Rules dialog box to change the rules for layers. You can enable or disable routing on any layer, and choose a preferred autorouting direction for any layer.

Choose the **Setup Routing > Layer Rules...** menu item to display the Setup Layer Rules dialog box.

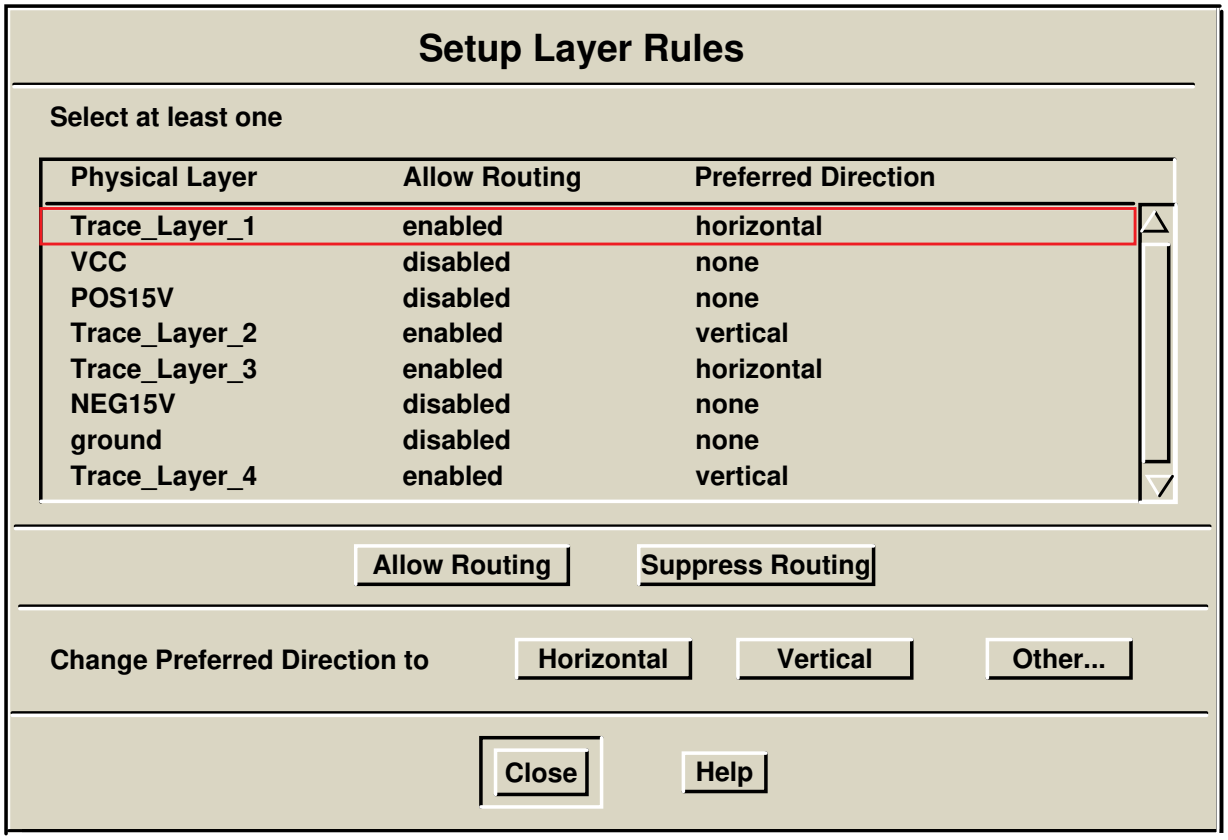


Figure 1-18. Setup Layer Rules Dialog Box

- **Allow/Suppress Routing**—enables and disables routing.
- **Horizontal**—sets the preferred direction for automatic routing to horizontal on the selected layers.
- **Vertical**—sets the preferred direction for automatic routing to vertical on the selected layers.
- **Other**—displays the Change Layer Preferred Direction dialog box that provides additional choices for preferred routing direction. A choice made in this box reflects in the Setup Layer Rules dialog box when you select **OK**.

Routing Attributes

The following attributes control routing:

- **Board_default_padstack**—associates a padstack with every component pin not specified with the Component_default_padstack or Component_padstack_override attribute. Required.
- **Board_routing_layers**—specifies the number of signal layers for your board. Required.
- **Board_routing_outline**—defines a polygonal area on the board inside of which routing is allowed. Required.
- **Default_pad_size**—specifies a pad diameter for use by the autorouter when calculating a non-uniform routing grid. Required.
- **Power_net_names**—specifies power net names for assignment to power planes. Required.
- **Diagonal_routing_allowed**—disables diagonal routing on the board if set to **no**; otherwise, diagonal routing is allowed by default and the attribute is not required. Optional.
- **Routing_keepout**—defines an area on the board in which routing is prohibited. This keepout applies to all signal layers by default, but can be limited to specific layers if desired. Optional.
- **Tjunctions_allowed**—disables routing of t-junctions on the board if set to **no**; otherwise, t-junctions are allowed by default and the attribute is not required. Optional.
- **Trace_keepout**—defines an area on the board in which traces are prohibited, but vias and area fills are allowed. This keepout applies to all signal layers by default, but can be limited to specific layers if desired. Optional.
- **Via_keepout**—defines an area on the board in which vias are prohibited, but traces and area fills are allowed. This keepout applies to all signal layers by default, but can be limited to specific layers if desired. Optional.

Routing Properties

The following properties control routing:

- **Net_length**—allows you to specify high and low limits on the length of a net. This property can be useful in high speed or critical timing situations. Length is measured in user units (the units you used in your board definition).

- **Net_type**—assigns a name to a net that is then associated with a set of routing design rules for all nets with that Net_type name. The rules are defined in the board technology design object under the net_type name.

- **Net_order**—specifies how the net must be broken up. The Net_order property value choices are: Starburst (default breakup), Daisy_chain, User, ECL, ECL_daisy_chain, and ECL_user.

- **Restrict**—defines special restrictions for a net. The Restrict property is useful where t-junctions are generally allowed, but specific nets must be restricted from having t-junctions.

Routing Design Rules by Net and Layer

The Net_type property allows you to assign design rules by net and by layer.

- To any nets on all layers.
- To any nets on specific layers.
- To specific nets on all layers.
- To specific nets on specific layers.

The value of the Net_type property is a name that corresponds to a named set of design rules stored with the design's technology data.

The named set of design rules has:

- A default set of net rules, which are rules for the net type.
- Layer net rules, which are an optional set of override rules for specific layers for the net type.

All nets that do not have a Net_type property are placed in a default net class that uses the net type name Default_net_type.

Net_type Property Design Rules

Numerous named sets of design rules, which establish values for any of the listed items, can be created.

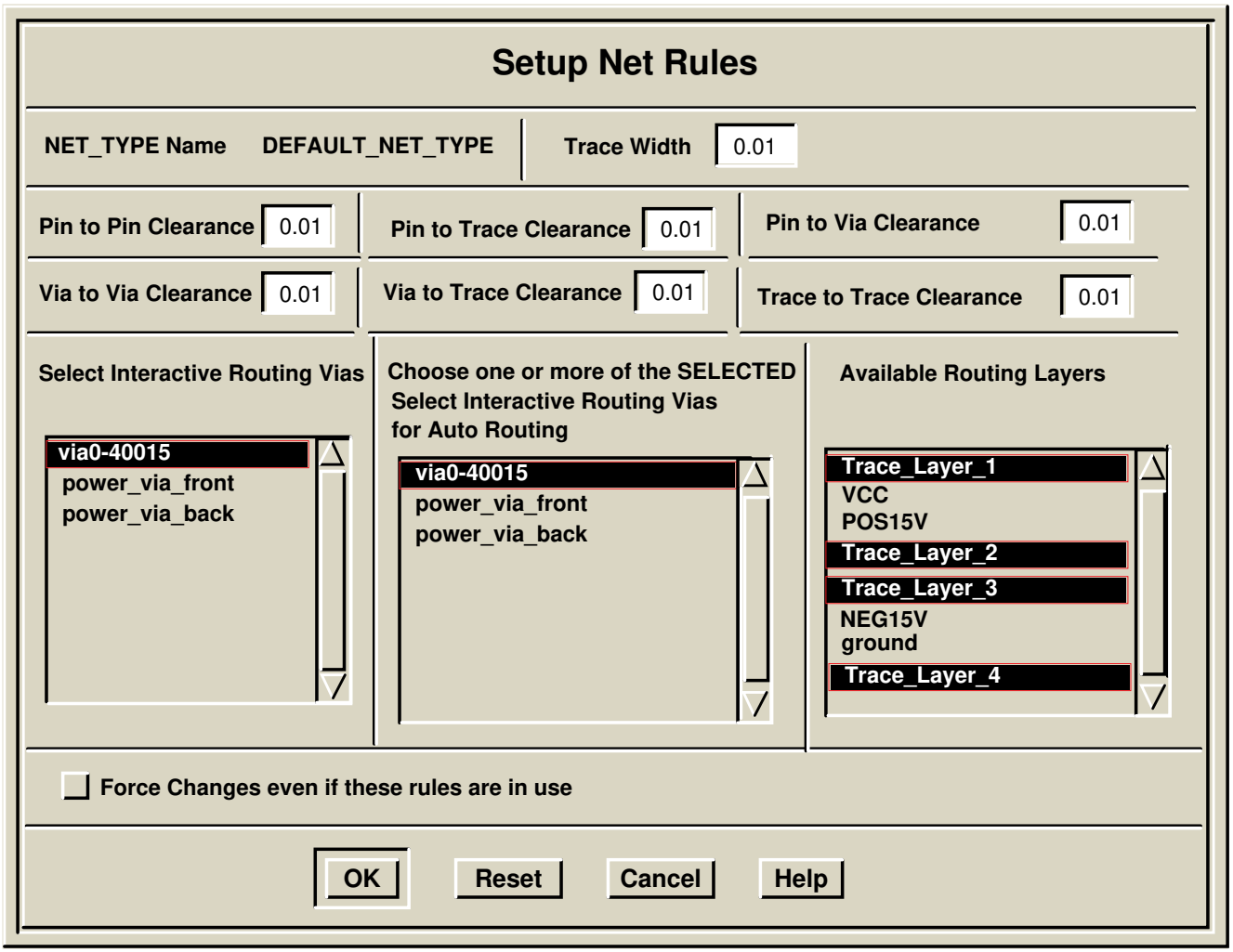


Figure 1-19. Setup Net Rules Dialog Box

You can associate these named sets of design rules with any net or combination of nets by assigning a Net_type property with the name of the rule set as a value. The name assigned to the set of design rules for nets having no Net_type property is DEFAULT_NET_TYPE.

Net Rules for Layers

Net rules for layers enable you to modify a net type's design rules on a layer-by-layer basis. You set the rules by selecting the **Setup Routing > Change Net Rules for Layers...** menu item.

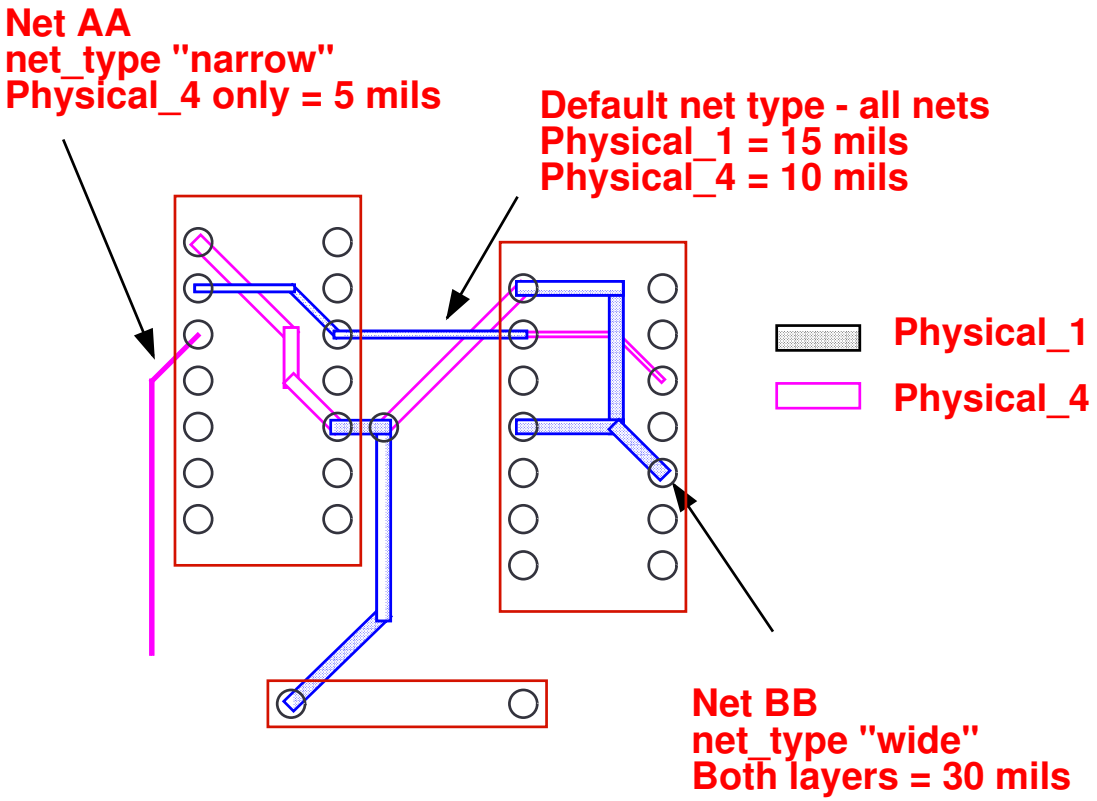


Figure 1-20. Net Rules Assigned by Layer

Here are some examples of net rules.

- Originally, a net type called default_net_type exists for all layers. In this example, the layers are Physical_1 and Physical_4. The default trace width for all nets on these two layers is set at 15 mils.
- A second net type called wide is defined, with the Setup Net Rules dialog box, in order to set some trace widths to 30 mils.

- A third net type called "narrow" is defined for 5 mil traces, but only Physical_4 is selected for routing.

  The menu item **Setup Routing > Change Net Rules for Layers** was chosen to change the default net type trace width to 10 mils for traces on Physical_4 (traces on Physical_1 remain at 15 mils).

If the wide net type is assigned to net BB and the narrow net type is assigned to net AA, you could expect the following results:

- Net BB routes at 30 mils on both layers.
- Net AA routes at 5 mils, but routes only on layer Physical_4.
- All other nets route at 15 mils on layer Physical_1 and at 10 mils on layer Physical_4.

Interactive Routing Features

You can set the following features to control interactive routing.

- **All angle routing**—LAYOUT provides the capability to interactively route traces at any angle. In order to use this feature, you must enable it with a menu selection. All angle routing then remains available until you disable it with another menu selection.

- **Routing via**—You can specify a via to use for interactive routing. If you do not make a choice, the interactive router uses the first via listed in the list of available vias for the net type currently being routed. If you do make a choice, but the specified via is not among the available vias for the net type the interactive router is routing, the router uses the first available via for that net type. Available vias are listed in the net rules for each net type.

- **Snap direction**—Vertex snap direction determines the allowable directions in which vertices can be entered during interactive routing, regardless of whether cursor snap is on or off.

- **Snap to autorouter grid**—If you combine interactive routing with automatic routing, it is advisable to route interactively on the same grid used by the autorouter. This is particularly useful if you are routing with a non-uniform grid or a pattern grid.

- **Cursor snap to display grid**—The display grid is the grid of dots displayed on the screen. If you set it to the same values as the uniform grid on which you want to route, you can then set the cursor to snap to the grid and be sure that your traces are entered on the routing grid.

Interactive Routing Environment

You can set the following features to control your display during routing.

- **Display of guides**—Guides illustrate unrouted connections. You can use guides when routing interactively. When you route a connection, the guide is replaced with the resulting trace. When you delete a trace, a guide replaces it.

- **Display of traces**—You can affect the display of traces, pads, and vias in two ways. First, you can specify how traces and polygons are drawn. For example, they can be drawn as centerlines, as outlines, or as polygons filled with a solid color or pattern. Second, you can choose to have the edit layer (the layer on which you are routing or editing traces) drawn on top of other displayed layers, making the active items more clearly visible.

- **Display of reference grid**—The reference grid is the grid of dots displayed on the screen. It is useful as a reference for estimating distances, and can be a useful routing aid if set to the same X and Y values as the uniform grid on which you want to route.

Routing a Guide

Routing a guide is a simple process of selecting the guide and then adding vertices to create trace segments along the desired path until the connection is complete. If you stop routing before the trace is complete, a guide remains between the point where you stop and the pin or t-junction that defines the endpoint of the guide. To route a guide, proceed as follows:

1. In the Edit window, choose the **Routing > Route Interactive** popup menu item.

Figure 1-21. Route Interactive Prompt Bar

The Route Interactive prompt bar offers the following choices:

- location button—changes the cursor to a large crosshair and enables the Select mouse button to enter vertices at the cursor location.
- first stepper button—provides choices to limit routing element selection or create a new guide:

  guide or trace—enables selection of whichever is closer to the cursor location when the Select mouse button is clicked initially to select a routing element.

  guide—enables selection of the guide closest to the cursor location when the Select mouse button is clicked initially to select a routing element.

  new guide—enables creation of a new guide between the cursor location (a pin or trace) and another pin in the same net when the Select mouse button is clicked initially to select a routing element.

- second stepper button—disables the repeating operation of the prompt bar. It is recommended that you not change the default setting of this button.

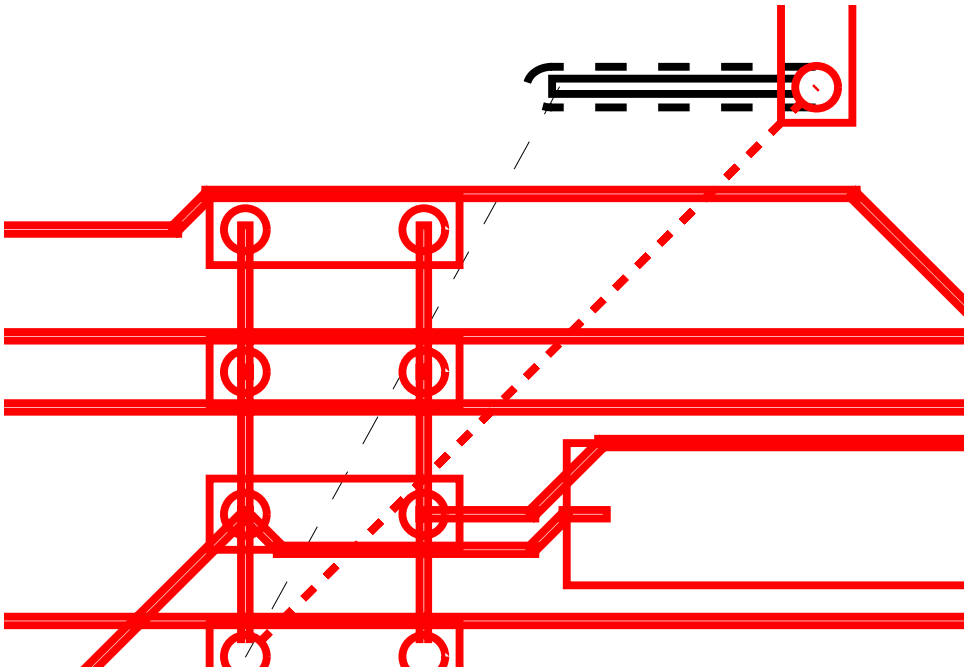


Figure 1-22. Interactive Routing

2. Place the cursor over the guide you want to route and click the Select mouse button.

   The guide is selected, the cursor changes to a small arrow, and a dynamic graphic called a trace drag is drawn. The drag continuously shows both the segment that would be created between the initial and current cursor locations and the guide that would remain after insertion of a vertex at the current cursor location. If the current view style is outline or filled, the drag depicts the full trace width and current trace-to-trace clearance.

3. Move the cursor to the location of the first desired bend and click the Select mouse button.

   A vertex is entered at the cursor location and a trace segment is created. The guide remains selected, but is now drawn between the new vertex and the guide endpoint. A new trace drag begins from the end of the segment just entered.

4. Continue to create trace segments in this fashion until the entire connection can be completed with the addition of just one more segment.

5. To complete the connection, place the cursor over the endpoint of the remaining guide and click the Select mouse button.

   The final trace segment is created, the cursor changes to a large crosshair, and the message, `"Connection is complete"`, displays in the Transcript window.

   The prompt bar remains and the Select mouse button is enabled to select a new routing element to begin another series of routing steps. To terminate interactive routing, select **Cancel** in the prompt bar, or press the ESC key.

Interactive Routing Guidelines

Interactive routing is primarily controlled by two things:

- Layers set as toggle layers
- Via rules

Toggle Layers

The Edit Layers dialog box lists all the layers in your design. The layers selected as toggle layers become active for the purpose of interactive routing.

1. To set up toggle layers for interactive routing, you select menu item **Setup Routing > Edit Layer**.
2. In the Setup Edit Layer dialog box, you click the Select mouse button on a layer name to select that layer as a toggle layer for interactive routing.

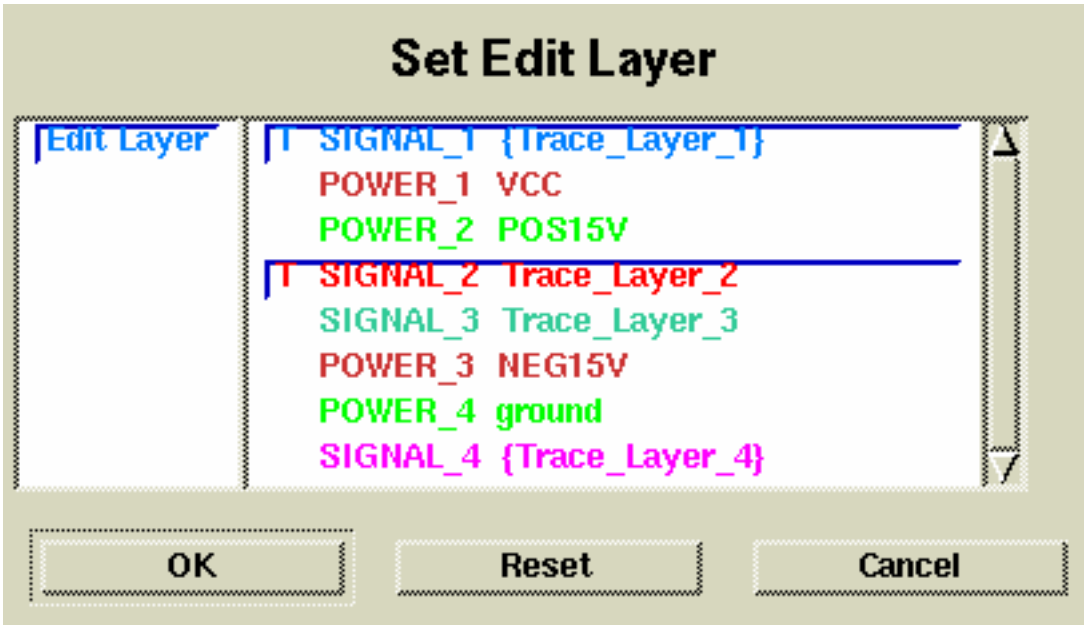


Figure 1-23. Setup Edit Layer Dialog Box

When a layer is selected as a toggle layer, the layer name has a box around it and a T appears to the left of the layer name. The Edit Layers dialog box above shows that layers Signal_1 and Signal_2 are selected as toggle layers. You can interactively route between layers Signal_1 and Signal_2.

3. To set one of the toggle layers as the current Edit layer on which to start interactive routing, click the Select mouse button in the area to the left of the layer name. The label *Edit Layer* appears next to the layer you select.

   The Edit Layer Display Area displays the current Edit Layer, as shown below.

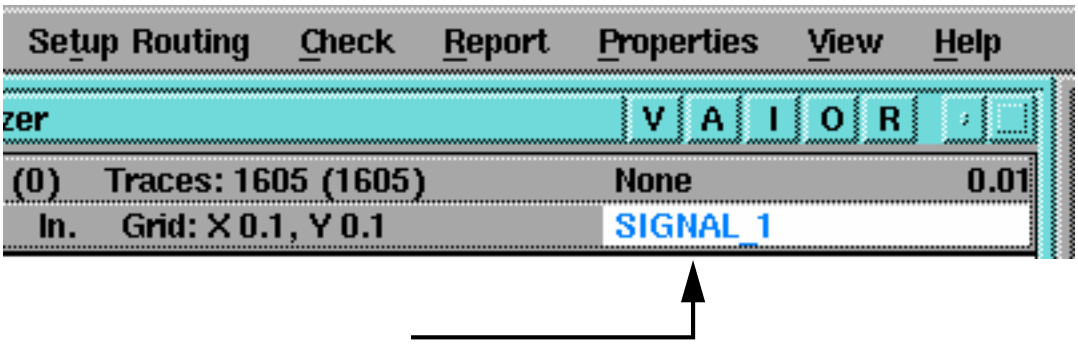


Figure 1-24. Edit Layer Display Area

Via Rules

The rules associated with the vias in your design determine which via is appropriate to use when routing specific nets.

Suppose you are interactively routing between layers Signal_1 and Signal_2. You have a via called Via040015. Via040015 has the following start and stop via rules associated with it.

- Signal_1 to Signal_2
- Signal_2 to Signal_3
- Signal_3 to Signal_4

Because the via has a rule that goes from Signal_1 to Signal_2, Via040015 can be used to interactively route with Signal_1 and Signal_2 set as the toggle layers.

Choosing an Interactive Routing Via

The net rules for the net type determine the default via for interactive routing. You define the via or vias each net type uses when you define the net rules.

You set up net rules for each net type using the Setup Net Rules dialog box. You set rules for the default net type, as well as for net types you define using the Net_type property. A net uses the default net type if it has no Net_type property associated it.

The default net type in this example has Via040015 set as the interactive routing via. When you route a net that uses the default net type, Via040015 is selected as the interactive routing via.

Routing to Power Layers

Suppose the guide you select to interactively route is a ground net. The ground net has a Net_type property attached that you defined as Power_nets. One of the vias used by the Power_nets net type is Via_power_front. Via_power_front connects from the top trace layer of the board to all the power layers. It has the following via rules associated with it.

- Signal_1 to Power_1
- Signal_1 to Power_2
- Signal_1 to Power_3
- Signal_1 to Power_4

When you select the guide of a net using the Power_nets net type, the system selects the first via in the Power_nets interactive routing via list as the interactive routing via. In this case, Via_power_front is the interactive routing via.

To connect to the ground layer, you must also select the ground layer as a toggle layer in the Setup Edit Layers dialog box. The dialog box below shows that the ground layer is Power_4.

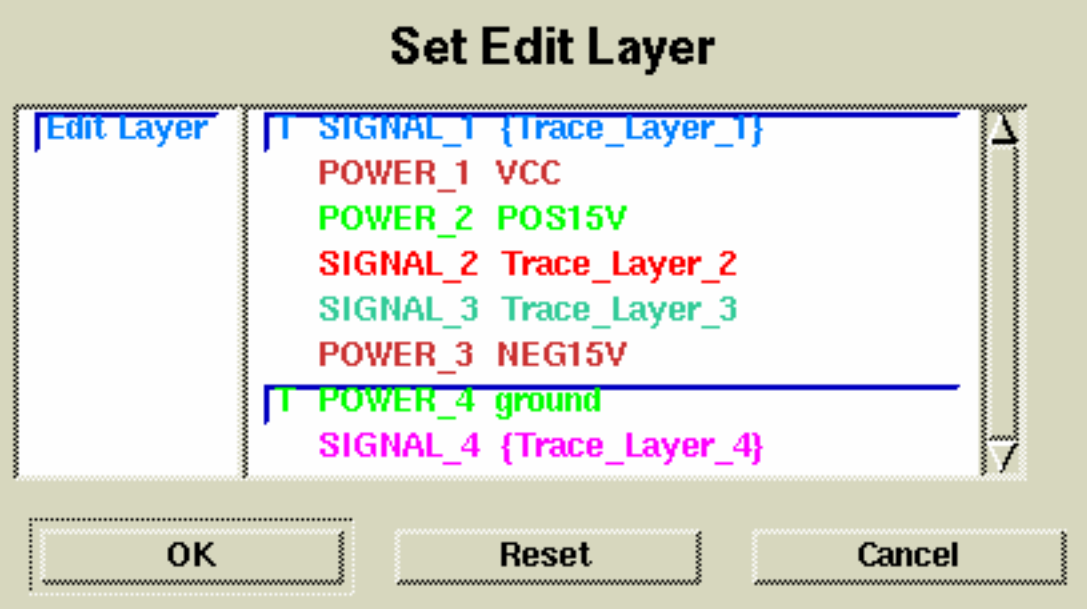


Figure 1-25. Trace_Layer_1 and Ground Set as Edit Layers

To add the ground layer to the list of toggle layers:

1. Select menu item **Setup Routing > Edit Layer.**
2. Click the Select mouse button on layer Power_4.
3. **OK** the dialog box.

Hiding Guides

It might be helpful to hide the guides of power nets to unclutter the view. The **View > Hide Guides** menu item displays the Hide Guides dialog box.

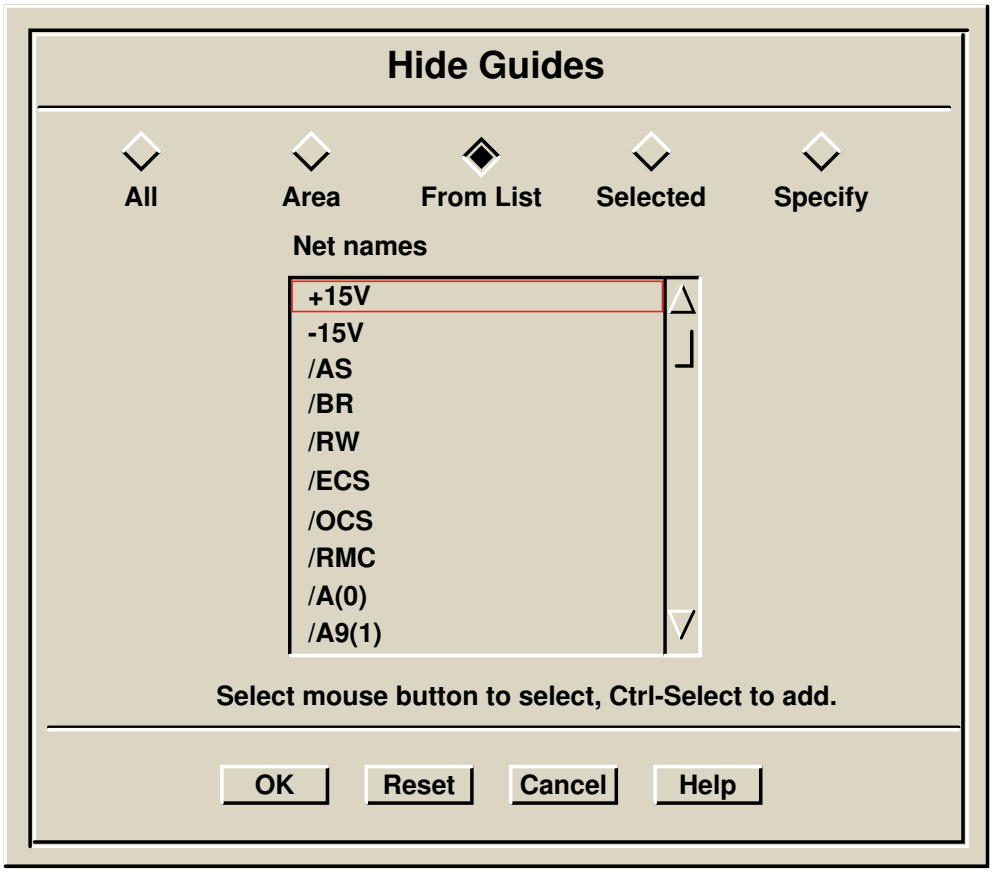


Figure 1-26. Hide Guides Dialog Box

Selecting **From List** displays a list of the nets in the design. You can select the net for which to hide guides.

Protecting Routing

Because the autorouter is a ripup and re-route router, any existing routing is not protected from the autorouter.

Protected routing cannot be ripped up and rerouted by the autorouter. Use **Routing > Protect > Traces** to protect selected traces, vias, and area fills.

You can also protect entire nets. Use **Routing > Protect > Nets** to protect all the routing elements in specified nets.

Protected routing cannot be selected until you unprotect it, and it is not be disturbed by the autorouter.

Lab Exercise

In this lab exercise you interactively route a section of your design that contains analog circuitry. Because analog sections typically use a variety of trace widths and design rules, you create a new net rule for the analog section. Using the new net rule, you completely route the analog section of your design.

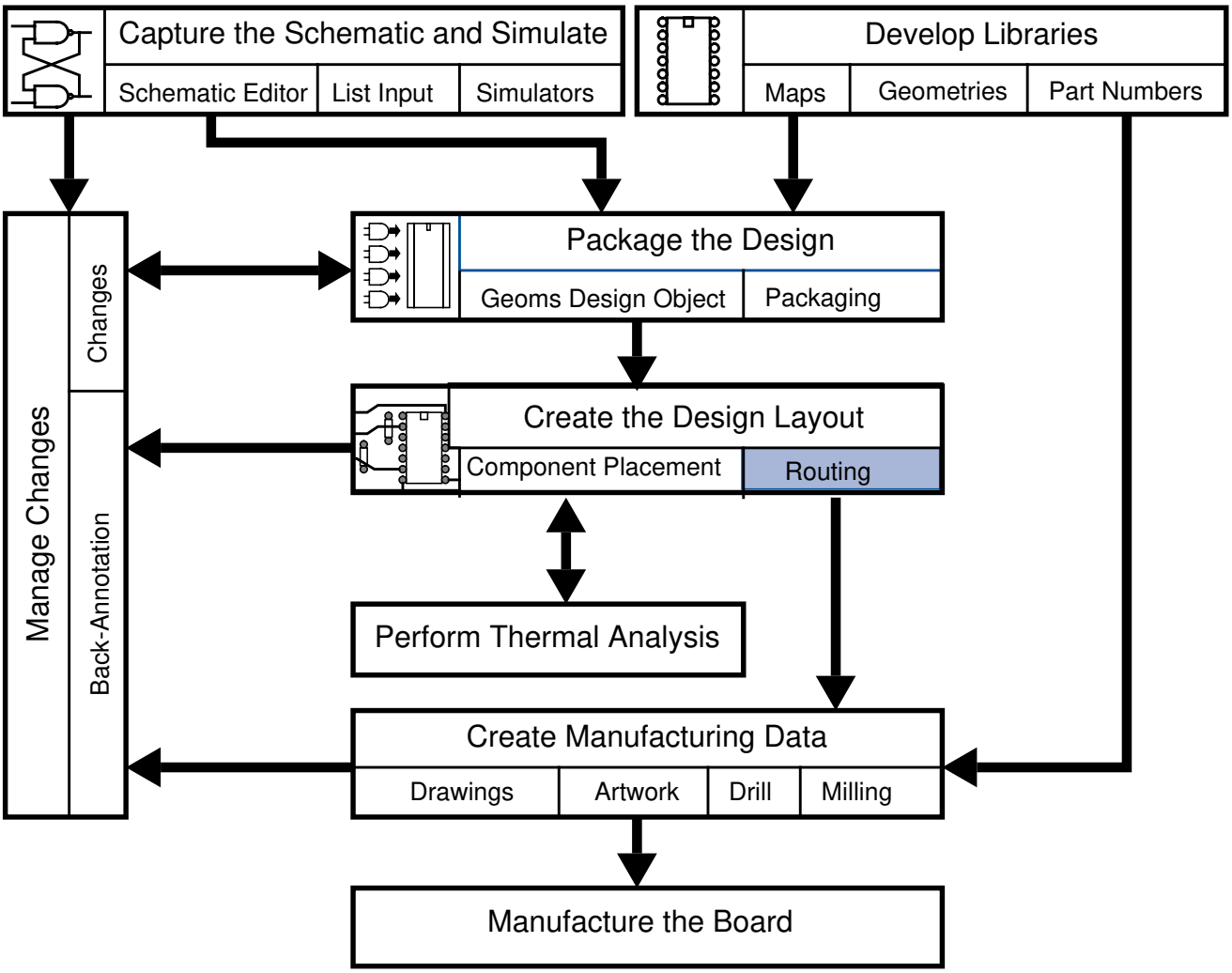
Upon completion of this lab exercise you are able to:

- Create new net rules.
- Assign net rules to nets in analog section of design.
- Interactively route nets in the analog section.
- Select and protect traces in the analog section.

Turn to Module 6—Lab 1: "Design Rules and Interactive Routing".

Lab 1
Design Rules and Interactive Routing

Routing Traces on the circuit board is the second part of the discussion on the circuit board design process. Lab 1 familiarizes you with setting up for routing, as well as interactively routing your design.



Introduction

In this lab exercise, you interactively route a section of your design that contains analog circuitry. Because analog sections typically use a variety of trace widths and design rules, you create a new net rule for the +15V and -15V nets in the analog section. Using the new net rule, you completely route the analog section of your design.

Upon completion of this lab exercise you are able to:

- Create new net rules.
- Assign net rules on a per-layer basis.
- Interactively route nets in the analog section.
- Select and protect traces in the analog section.

In this module, you see pictures of palette icons that correspond to menu paths and pictures of strokes that correspond to menu selections. Try using the icons in the palette menu and the strokes where indicated. They are quick and easy methods of accessing functions.

Procedure

You interactively route a section of your design that contains analog circuitry, create a new net rule for the +15V and -15V nets in the analog section, and then completely route the analog section of your design using the new net rule.

Preparation for Lab

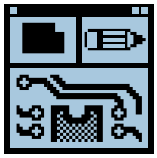
To set up for this lab, invoke the LAYOUT tool on your design.

1. If you or your instructor have not already done so, complete the Installation Procedure in the About This Training section of this manual.

2. Invoke the Design Manager by entering the following in a shell:

```
$MGC_HOME/bin/dmgr
```

3. Using the Design Manager, change your current directory to the board_new directory by clicking on the four-way icon in the navigator window. In the "Change directory to" dialog box, enter the pathname: your_path/**training/board_new/mod6** and press the Return key.



layout

4. Find the LAYOUT icon in the Tools window. Invoke LAYOUT by placing the cursor on the LAYOUT icon and double clicking the Select mouse button.

   The INVOKING LAYOUT: Select a Design dialog box displays.

5. Select the **sig_az** design and **OK** the dialog box.

   A Report-Startup message appears in the middle of the LAYOUT Session window. This report is a list of notes concerning the files used to invoke the LAYOUT tool.

6. Look over the report, close the report window, and maximize the size of the LAYOUT session window to fill the display.

Enlarging the Transcript Window

Increasing the number of lines shown in the transcript window helps you read messages output from LAYOUT.

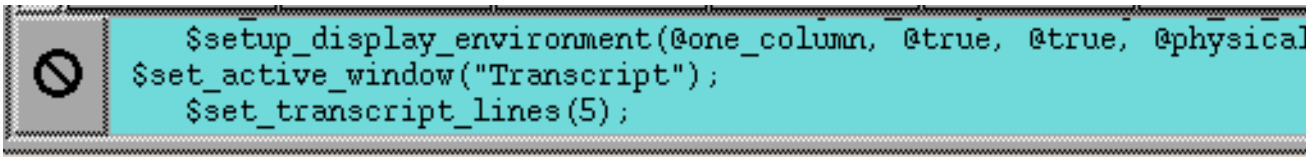


Figure 1-27. Transcript Window

1. Place the cursor in the transcript window.
2. Press and hold the Menu mouse button (the right mouse button by default) to display the menu.
3. Select the **Set Transcript Lines** menu item.

   The Set Transcript Lines prompt bar displays.

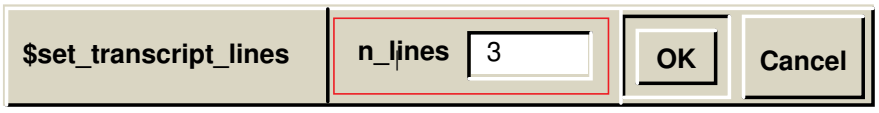


Figure 1-28. Set Transcript Lines Prompt Bar

4. Enter **5** the n_lines field and **OK** the prompt bar.

The Transcript window increases to display five lines of text.

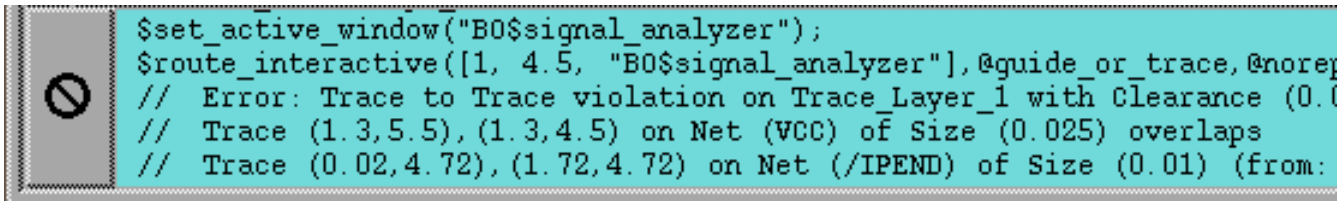


Figure 1-29. Enlarged Transcript Window

Setting Up Design Rules

Before beginning interactive routing, you need to set up some conditions.

Set up Routing Rules

Choose the **Setup Routing > Routing Rules...** menu item to set up your routing rules for interactive routing.

Fill in the dialog box as follows, then **OK** the dialog box.

It is not necessary to fill in the Y values, as they are the same as those you assign to the X direction.

This is the default routing grid.

Trace Grid X: **0.025**
Pin Grid X: **0.1**
Via Grid X: **0.05**
T-junctions Allowed
Diagonals Allowed
Trace Vertex Bends: **Orthogonal and Diagonal**
Pad and Via Entry: **Orthogonal and Diagonal**
Route Connections By: **Length**

Set up Layer Rules

1. Choose the **Setup Routing > Layer Rules...** menu item to set up the layer rules for interactive routing.

   By default, the layers are most likely set as specified here. It is always best to check and ensure the settings are correct.

2. In the dialog box, enable routing on physical layers Trace_layer_1, Trace_layer_2, Trace_layer_3, and Trace_layer_4.

3. Also verify that the preferred routing direction is

   horizontal for Trace_layer_1
   vertical for Trace_layer_2
   horizontal for Trace_layer_3
   vertical for Trace_layer_4

4. When you finish, **Close** the dialog box.

Set up Net Rules for a New Net Type

Set up a new net type for the +15V and -15V nets.

1. Choose the **Setup Routing > Net Type Rules...** menu item to set up the routing net rules for the analog section. In the **Setup Net Type Rules** dialog box, choose **Add...**.

2. In the Add Net Rules dialog box, enter the following.

Interactive and autorouting respect these rules.

Net Type: **POWER_15**
Trace Width: **0.025**

Pin to Pin Clearance: **0.010**
Via to Via Clearance: **0.010**

Pin to Trace Clearance: **0.010**
Via to Trace Clearance: **0.010**

Pin to Via Clearance: **0.010**
Trace to Trace Clearance: **0.010**

Select Interactive Routing Vias: **[Select all vias that appear]**

Choose one or more of the SELECTED
Interactive Routing Vias for Auto Routing:
power_via_front
power_via_back

Available Routing Layers: **Trace_layer_1**
Trace_layer_2
Trace_layer_3
Trace_layer_4

To select more than one via or layer, choose the first item, hold down the CTRL key, and click the Select mouse button on the layer names.

3. **OK** the dialog box.

4. **Close** the Setup Net Rules dialog box.

Attach the New Net Type to Nets

1. Select nets +15V and -15V as follows.
   - Choose menu item **[Routing] Select > Select by Name**.
   - In the Select by Name dialog box, choose to Select By **Net** and select net names +15V and -15V from the list box.
   - **OK** the dialog box.

To select more than one net name, hold down the CTRL key, and click the Select mouse button on the net names.

2. Choose the **Properties > Net Properties > Net_Type...** menu item to assign the new net type to the nets.
3. In the dialog box, select the property **net_type**. Select net_type **POWER_15** and **OK** the dialog box.

   You have just assigned the POWER_15 net rule to each of the selected nets.
4. You can verify the property assignment by choosing menu item **Report > Selected**.

   The report shows the Net_type property and value for each net.
5. Close the report window and Unselect All.

   When you begin to route either interactively or with the autorouter, traces that are part of the +15V and -15V nets route with the specified width and clearance.

Altering Net Rules for a Specific Layer

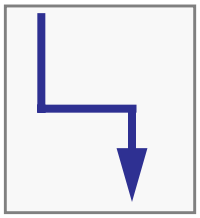
You can alter net rules for specific layers.

In the net rules, you previously set a trace width of 0.025 for the POWER_15 net type. Let us suppose you decide to use a trace width of 0.020 for the +15V and -15V nets when they route on Trace_layer_1. However, you want to use the 0.025 trace width on the other routing layers. You can set alternate net rules on a per-layer basis.

1. Choose menu item **Setup Routing > Net Type Layer Rules**.
2. In the Setup Net Type Layer Rules dialog box, select **POWER_15**, then select **Change...**.
3. In the Change Net Type Layer Rules dialog box, select the entries for **Trace_layer_1**, then select **Change...**.
4. In the Change Net Type Layer Rules dialog box, change the Trace Width to **0.020**, then **OK** the dialog box. The change reflects in the Change Net Type Layer Rules dialog box.
5. **Close** the dialog box. Also **Close** the Setup Net Type Layer Rules dialog box.

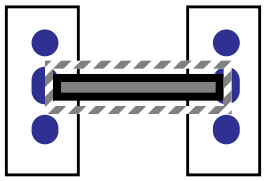
When you route traces having the POWER_15 net type attached, you use a 0.020 width trace on Trace_layer_1 and a 0.025 width trace on the other trace layers.

Setting the Display



1. Use the stroke shown at left to display the Display Controls dialog box.

   - Ensure the **Guides**, **Pads**, and **Vias** objects are enabled.
   - In the View Layers section, ensure the following layers are visible, then **OK** the View Layers dialog box: **Routing_keepout** and **Signal, Signal_1, Signal_2, Signal_3, Signal_4**.
   - In the Other Options section, choose Component Label.... Select to view the **Reference** designator. **OK** the View Component Labels dialog box.
   - In the View Style section, select **Filled** as the Polygon/Text/Trace style.



When you see a ghost image of a trace when routing, you see the actual size of the trace, and you also see the extent of the net rules. Because the polygons are now filled, it is easier to see the pins of the components, and the placement rule areas.

If later you decide that the filled polygons and traces appear too cluttered for your taste, you can choose a Polygon/Text/Trace style of Outline. Set a width override so you can see the connectors more clearly.

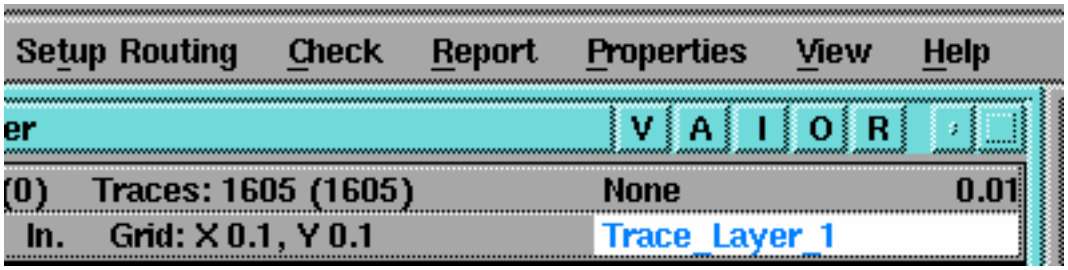
   - Choose the **Width Override** button. In the **Set Object Width Override** dialog box, set an override of **1**. **OK** the dialog box.

2. **OK** the Display Controls dialog box.

3. Choose the **Setup > Grid...** menu item. Set the display grid to **0.025** and the display interval to **2**. **OK** the dialog box.

   You are now ready to begin interactively routing the analog section.

4. Choose the **Setup > Display Environment** menu item. Unselect the Status Layer to Logical button. **OK** the dialog box.



The Edit Layer display now reflects the physical layer name.

Set up the Edit Layers

For interactive routing, you set up the Edit Layers similarly to setting the available routing layers for net rules. To set up the Edit Layers, use the palette menu on the right side of the session window.

1. Select the Route palette. Then select the **Edit Layer** icon.

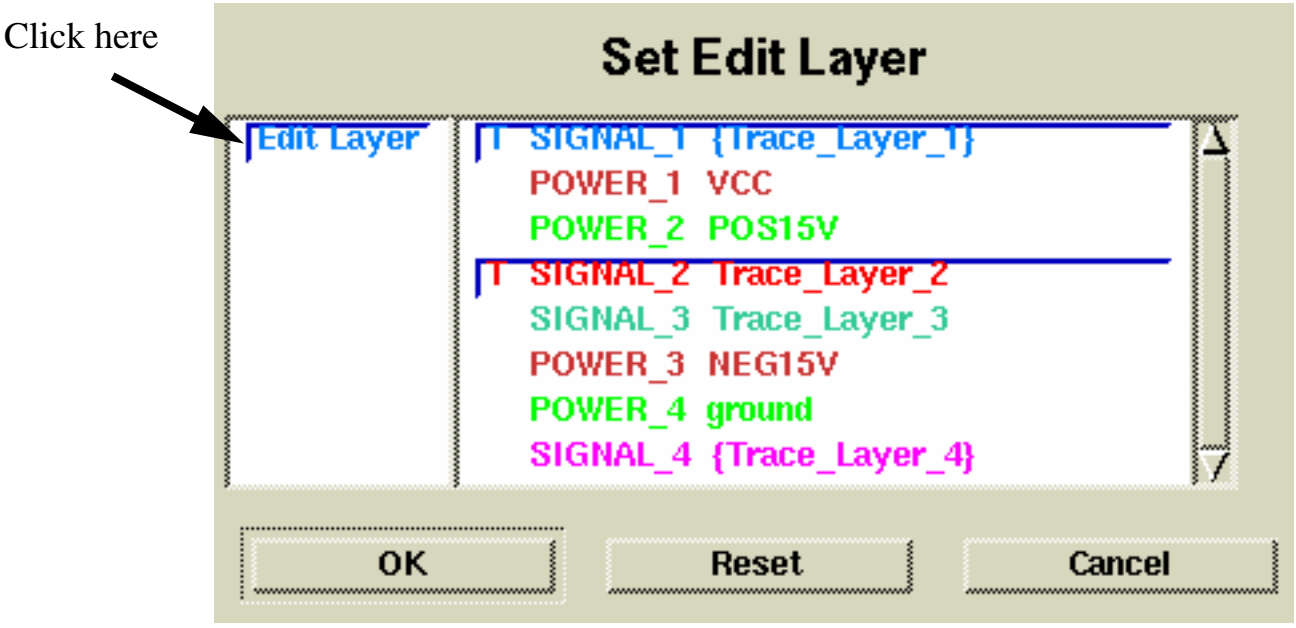


Figure 1-30. Set Edit Layer Dialog Box

2. In the Set Edit Layers dialog box, ensure only layers

   Trace_layer_1

   Trace_layer_2

   are selected as Edit layers, as shown in Figure 1-30. When a layer is selected, it has a rectangle around the layer name.

3. In the area to the left of the layer names, click the Select mouse button next to Trace_layer_1. This makes Trace_layer_1 the current Edit layer.

4. **OK** the dialog box.

Performing Interactive Routing

In this section, you interactively route the analog section of the board.

1. Use the View Area stroke to view the area around the analog section of the board so you can see it more clearly.

2. In the Edit window, choose the **Routing > Route Interactive** menu item.

   The Route Interactive prompt bar displays and prompts you for a location on a guide or trace. A guide is any one of the green (default color) lines that show a pin-to-pin connection. You replace guides with signal traces. Traces do not follow the same path as the guides. You must route the traces around other traces. Sometimes, you have to use vias to route around an existing trace.

3. Position the cursor on the guide connecting U69, pin 2 to the pin on U80. This guide is highlighted in Figure 1-31. Click the Select mouse button.

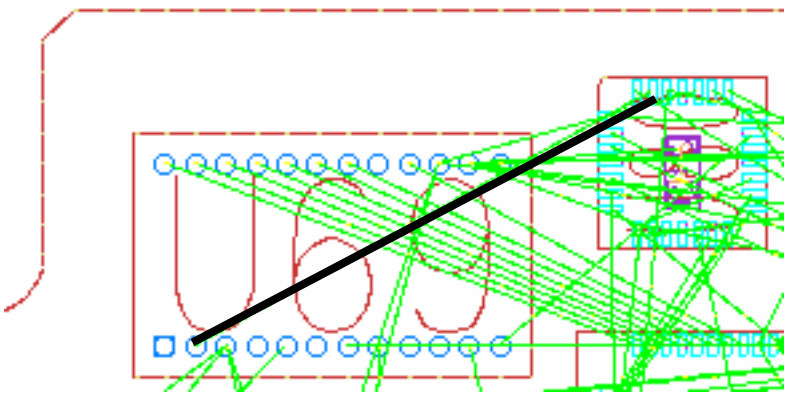
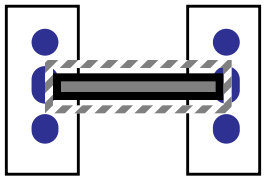


Figure 1-31. Guide to Interactively Route

To display pin numbers, set the Pin ID style to Physical in the Display Controls dialog box.

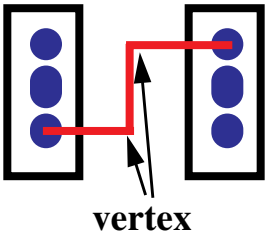
When you click on a guide, it becomes selected and highlights. A ghost image of the trace becomes visible. Via sites are also part of this ghost image. The ghost image follows the cursor movements beginning from the pin connection of the guide that is closest to where you clicked to select the guide. One end of the ghost image moves with the cursor, and the other end is tied to the nearest connected pin.



The ghost image appears as two sets of lines. The inner lines represent the actual size of the trace. The outer lines represent the extent of the net rules.

4. Move the cursor so the image of the trace roughly follows the direction of the guide, so you create the trace towards the other endpoint. The outer lines are not allowed to violate the net rules by touching a via or another trace or pin; if they do, you must move the cursor to reposition the ghost image before you can place the trace.

5. Click the Select mouse button.



A trace segment appears. You have created a vertex in the trace from which you can continue the trace in another direction. Also, a vertex is a location where you can create a via. Each time you click the Select mouse button, you add another vertex to the trace.

6. Continue routing the trace. Add vertices where needed. When the last point you click on is the terminating pin of the connection, the guide is removed, the trace is created and unselected, and you can route another trace.

   A message appears in the message window to verify the connection is complete.

Interactive Routing Techniques

Try the following techniques as you perform interactive routing on the board. Route the traces in the analog section. In addition, you can perform interactive routing in other areas of the board.

It is not necessary to route the entire analog section; just do enough routing to become comfortable with the techniques in this lab exercise.

Changing Edit Layers

When routing, you might need to drop a via to another layer to avoid pads and traces and to continue routing. When selecting a via for interactive routing, LAYOUT looks for a net type on the net you are routing. If no Net_type property exists on that net, LAYOUT looks at the default net type. LAYOUT selects the first via in the via list for the net type as the interactive routing via. To drop a via, you need the following:

- The appropriate via selected to connect the layers as shown in Figure 1-32.

Figure 1-32. Interactive Routing Via Via040015

- The routing layer enabled in the Edit Layer list, Figure 1-33, shows layers Trace_Layer_1 and Trace_Layer_2 set as the toggle edit layers.

Figure 1-33. Toggle Layers Trace_Layer_1 and Trace_Layer_2

Use the following sequence as a guide to dropping a via and changing layers while interactively routing.

1. If the Route Interactive prompt bar is not still visible, click on the Route Interactive icon in the Route palette.

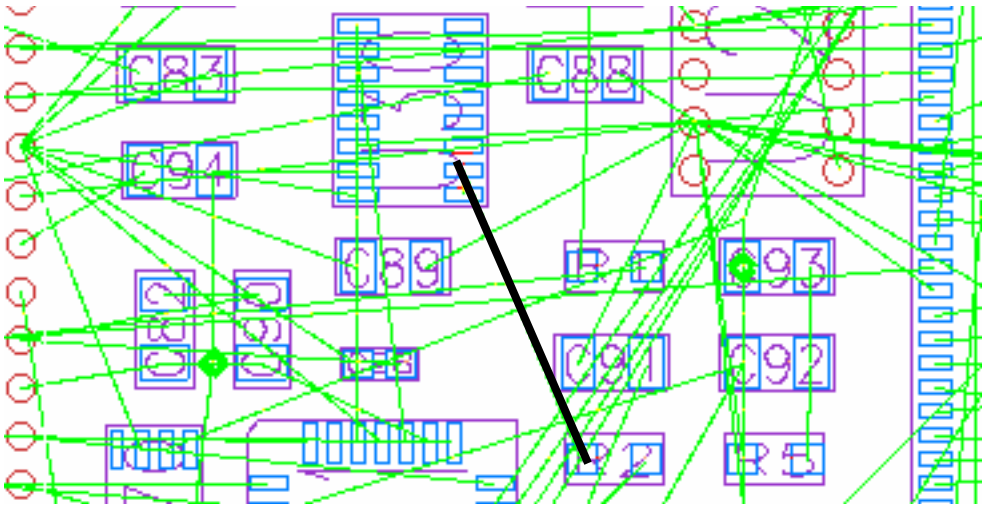


Figure 1-34. Guide to Route for Changing Layers

2. Click on the guide connecting the surface pad on component R2 to the surface pad on U30, as shown in Figure 1-34. Note in the upper-right corner of the screen that via040015 is set as the interactive routing via.

If you cannot read the reference designator text, turn off the Guide object. Find the components, then turn visibility of the Guide object back on.

3. Move the cursor and click the Select mouse button to put in trace segments.

4. When you wish to drop a via to another layer, select the Edit Layer icon in the palette menu. If the layer on which you want to continue routing is not already selected as a toggle layer, select the layer name in the dialog box. **OK** the dialog box.

5. To create the via, place the cursor on the last vertex you entered and click the Select mouse button a second time at this same location. Clicking twice at the same coordinates creates the via. The next toggle layer becomes the new Edit layer and you can continue routing the net.

Instead of setting only layer pairs as toggle layers, you can enable several or all layers listed in the Set Edit Layer dialog box. When dropping a via, you click the left mouse button in the same location as the last vertex once for each layer you want to move down the list of currently enabled toggle layers. You see the Edit Layer display area reflect each layer name as you click through the list. Stop clicking when you reach the layer on which you want to continue routing. As long as the selected via has a rule that connects to that layer, you can continue routing the net. If you toggle back to the layer on which you started, the via disappears.

If you try to connect a trace to an off-grid pin, it can be easiesr to click on the **Finish Trace** icon in the Route palette. Finish trace automatically adds a trace segment from the last vertex to the connecting pin.

Connecting to a Power Layer

Traces might not need to connect the two pins indicated by the guides. For example, some guides show connection between two pins, one of which is a surface mount pin connected to a power or ground plane.

A through pin connected to a power plane does not show a guide to the plane. Because the through pin goes all the way through the board, it automatically has a connection.

To connect to a power or ground layer, you start a trace from the surface pin, then create a via. The via connects the trace to the power or ground layer. You can determine if the selected guide is a ground or power connection by looking at the select count in the Status window. If the power or ground name displays in the window, you can finish the connection by creating a via.

To connect to a power layer, the power layer must be an enabled toggle layer in the Set Edit Layer dialog box.

To create a via, place the cursor on the last vertex you entered and click the Select mouse button a second time at this same location. Clicking twice at the same coordinates creates the via and completes the connection to the ground or power layer.

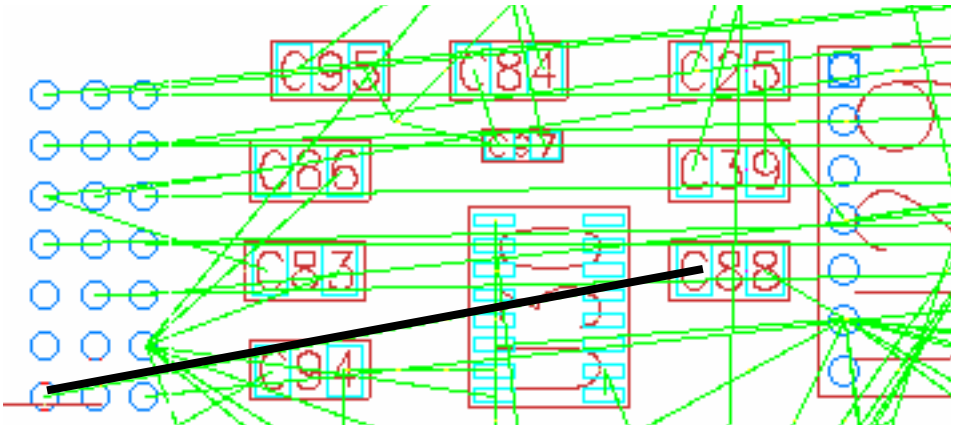


Figure 1-35. A VCC Net

Route a VCC net. Before you begin, ensure that layers Trace_Layer_1 and VCC are enabled as toggle layers.

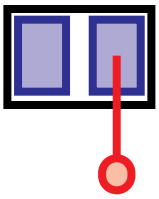
1. If the Route Interactive prompt bar is not still visible, select the Route Interactive palette icon.

2. Click the Select mouse button on the guide connecting component C88 to the connector, as shown in Figure 1-35.

   Notice that you have a select count of 2 and the net is VCC. The interactive routing via is power_via_front.

3. Begin the trace from the pad on component C88.

   If the trace begins from the connector, select the Other End palette icon.



4. Route out from the pad and drop a via to the VCC layer.

The via completes the connection. The guide to the pin on the connector disappears, because the connector pin is a through pin that already connects to the VCC layer. The pad on C88 is a surface pad. The via connects the surface pad to the VCC layer.

Trying Routing Techniques

The following routing techniques are given to you without specific instructions. As you route the analog section of the board, try out each technique. The idea is to become comfortable with interactive routing.

Routing to a T-junction

If two guides connect to a single pin, you do not have to route two traces all the way to the pin. You can route one trace to the pin, and then route the other trace so it joins with the first trace. As you route the analog section of the board, try this technique.

Finishing a Trace Automatically

You might discover a situation in which you attempt to complete a connection and the connection does not complete. The pin to which you are trying to route the trace might be on a surface mount component that is not on a grid point. Some of the surface mount components are not placed on the grid. To complete these connections, use the **[Routing] Finish Trace** menu item while the trace is still selected.

Troubleshooting Vias

If you try to create a via, but no via goes in, there might not be enough space for the via at that location. The trace vertex needs to be in an open space between traces and components so the via does not violate design rules.

While experimenting with interactive routing, you can also use the through via named via_thru.

Leaving a Trace Partially Routed

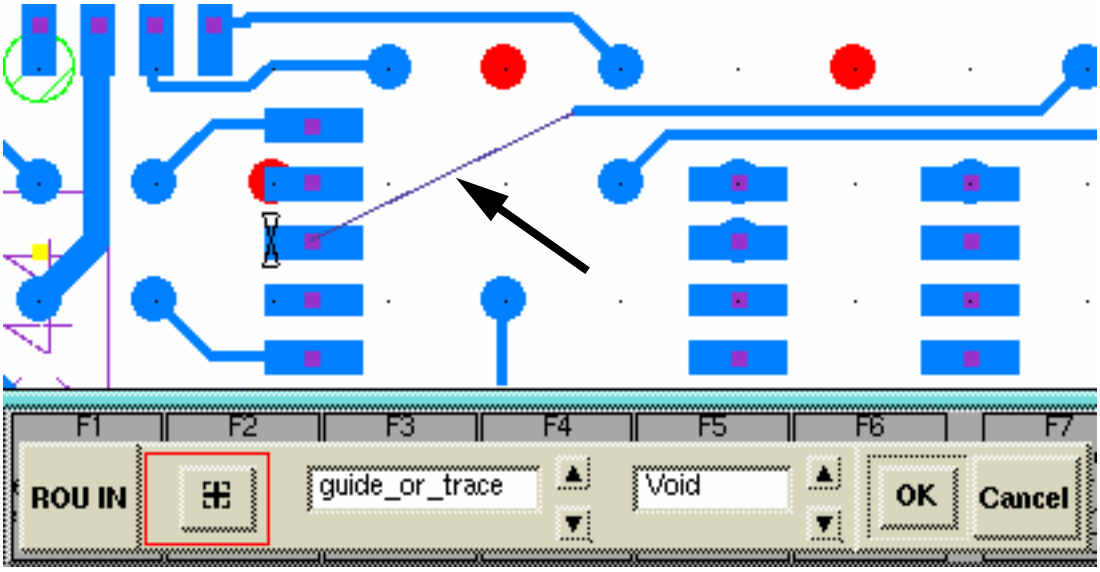
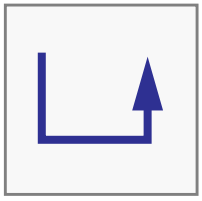


Figure 1-36. Unselect to Leave Trace Partially Routed



If you partially complete a trace and want to leave it temporarily to route another trace, use the Unselect All stroke. When the trace unselects, click the Select mouse button on another guide and begin routing the new trace.

Moving a Pin

Some geometries are defined with the Component_pins_moveable attribute, which allows you to move a pin once the component is placed on the board. A geometry with moveable pins allows you a degree of freedom in a tightly placed board. In this design, the dip14w geometry was defined with the Component_pins_moveable attribute. Move one or more of the pins to see how this feature works.

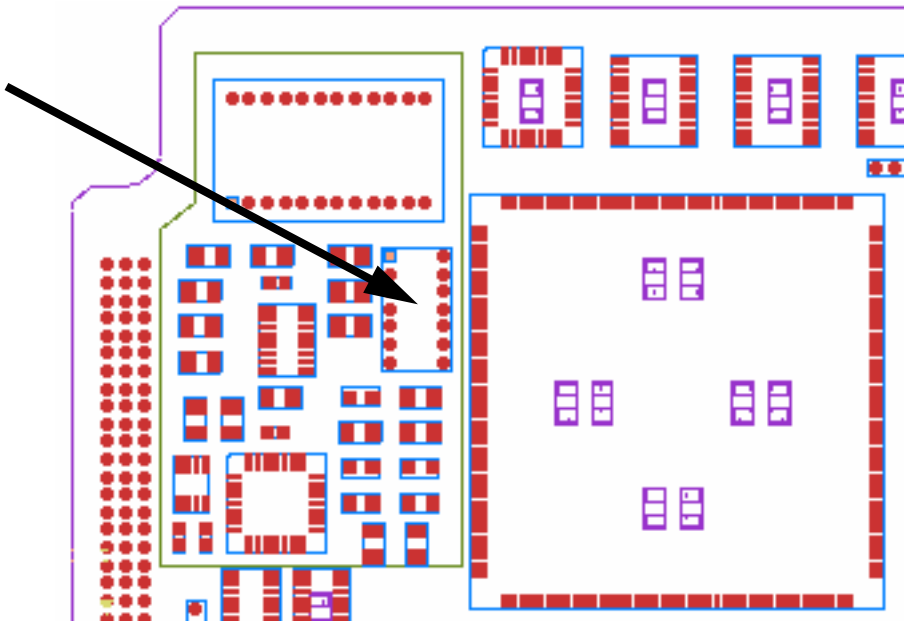
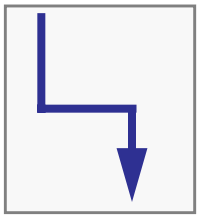


Figure 1-37. Geometry with Moveable Pins

1. If the Route Interactive prompt bar is still visible, **Cancel** it.



2. Use the Display Controls stroke to display the dialog box. Turn off the **Guides** object.

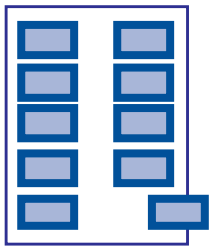
3. View an area around the analog section of the board shown in Figure 1-37.

   Component U29 uses geometry dip_14w. Dip_14w was created with the Component_pins_moveable attribute. U29 is the component to which the arrow points in Figure 1-37.

4. In the Edit window, select **[Routing] Extended Menu > Move Pin**.

   The prompt bar appears and the cursor turns into a crosshair.

5. Click the Select mouse button on one of the pins of the component and then click the Select mouse button at the new location for the pin.



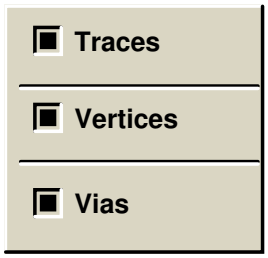
   The pin moves to the new location. You can move additional pins on the component is you like.

6. Try to move a pin on another component. You see an error message in the Message window. You cannot move pins on a component unless the Component_pins_moveable attribute was attached to the component geometry when it was created.

7. **Cancel** the prompt bar.

Moving a pin in this design is not necessary. However, it is good to understand this feature for designs where altering the location of a pin aids the routing.

Panning and Routing



1. From the menu bar, select menu item **Setup > Select Filter**. Set the filter so that only Traces, Vertices, and Vias can be selected.

2. Select **View > Display Controls**.

3. Select to view **Guides**, and select **Smart** guides.

   Unselect **Route Grid** to turn it off and Unselect **Edit Layer Only**. Leave other settings as they are. **OK** the dialog box.

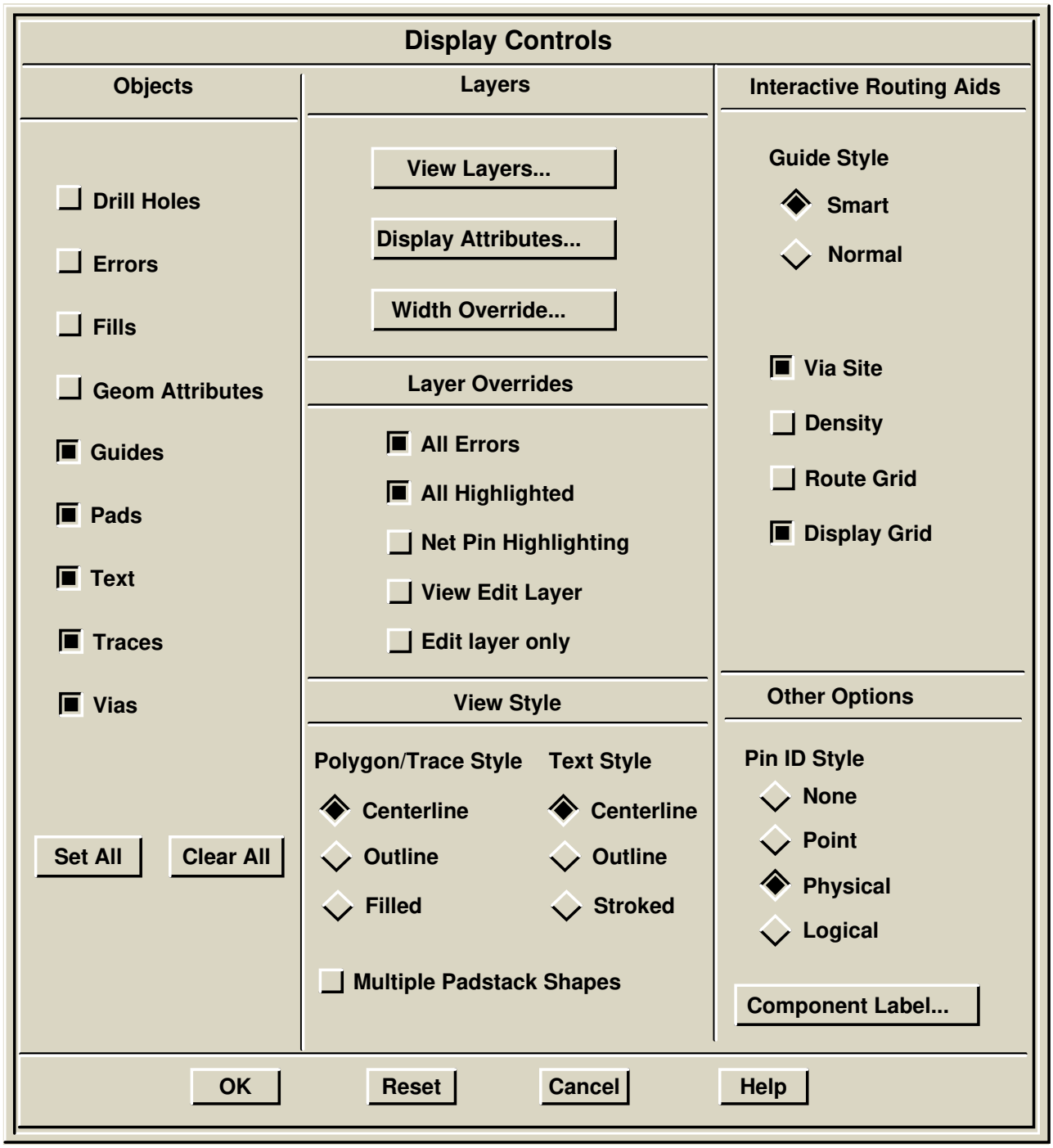


Figure 1-38. View Guides and Smart Guides

For the next few menu selections, use the Palette menu to the right of Edit window.

Figure 1-39. Palette Menu

4. From the top level Palette menu, click on **Route** to display the Route palette menu.

Now route one of these connections.

5. From the Palette menu, select **Route Interactive**.

6. Click the left mouse button on one of the guides. Choose a guide that appears to continue a good distance across the board.

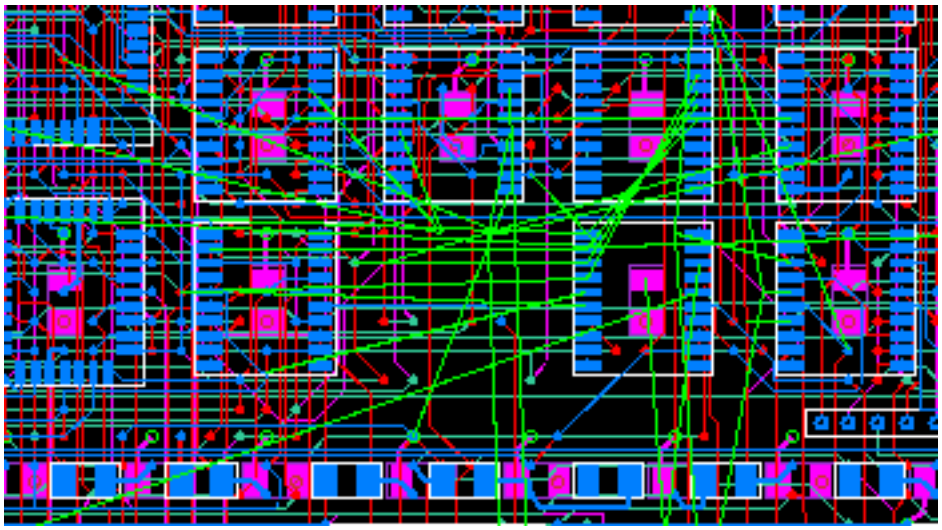


Figure 1-40. Select a Guide to Route

Because you have Smart guides set, guides not connected to the selected net disappear. Only guides of that net remain.

The trace begins from the pin closest to the point at which you clicked to select the guide.

7. Follow the guide by moving the cursor toward the next pin in the net. See how the pan slows and stops near the destination pin.

8. Move the cursor to keep on panning past the pin.

9. Complete the connection, if possible. **Cancel** the prompt bar.

Protecting Traces

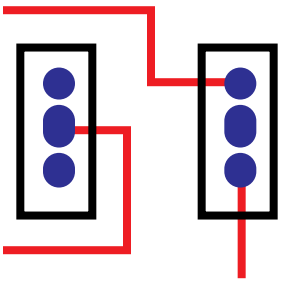
When you finish interactively routing traces, you need to protect them so they are not accidentally moved or deleted later. When traces are protected, you cannot select or change them. If you need to make adjustments, unprotect the traces.

1. Choose the **Setup > Select Filter** menu item. In the dialog box, specify to select only **Traces**, **Vertices**, **Vias**. **OK** the dialog box.

2. Choose the **[Routing] Select > Select All** menu item.

   All the traces are selected.

3. Choose the **[Routing] Protect Routing** menu item. Select **Protect Selected Routing** and **OK** the dialog box.



   The traces change color (to red by default) indicating they are protected. The automatic router does not rip up and reroute protected traces during the automatic routing stage.

   When you protect, highlight, or select an element, it changes color. You use the **View > Set Display Attributes...** menu item to change the color of protected elements on a per layer basis. Choose **Objects and Attributes**. Select the **Protect** option and change the color assignments for protected elements on the signal layers.

You can also access Set Display Attributes from the Display Controls dialog box.

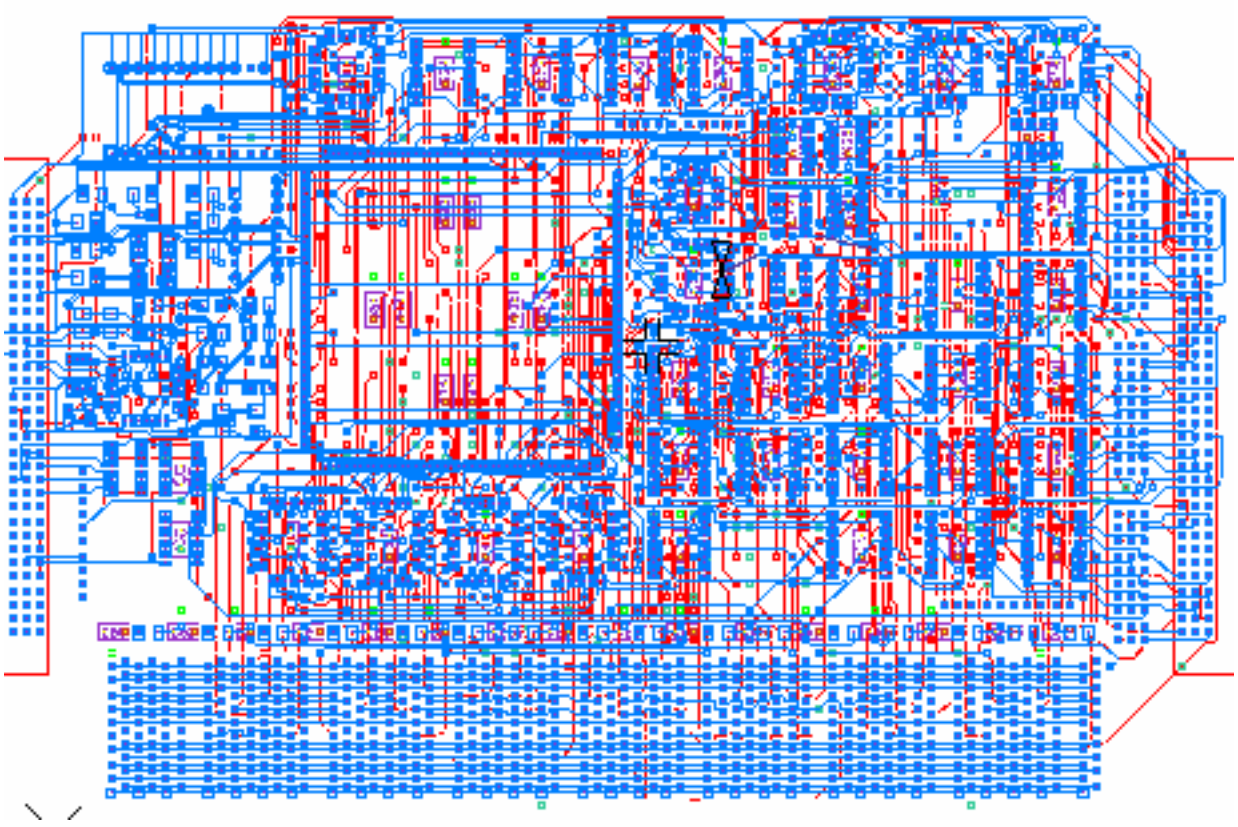
4. When you finish the lab, select menu item **File > Save > Design All** to save your work.

5. **Close** the LAYOUT session. It is not necessary to save your changes again.

   Congratulations! You have completed the "Design Rules and Interactive Routing" lab exercise. Continue with Lesson 2: "Automatic Routing".

Lesson 2
Automatic Routing

In the previous lesson, you learned how to set up for routing and how to perform interactive routing on your design. In this lesson, you learn how to automatically route the board.



Objectives

In this part of the routing module, you examine in detail the operation of the automatic router. To gain the best performance from the automatic router, you need an understanding of:

- Routing grids.
- Parameters that control the autorouter.
- Auto router operation.

Types of Grids

A routing grid creates the pattern the autorouter uses to route the board. Grid lines are the pathways the autorouter follows to route traces and make connections.

Auto router grids are one of the most important elements in a successful layout. In order for the auto router to provide optimal performance and route a design as completely and as fast as possible, the routing grid must be optimized for the mix of pin pitches, pad types, and component density on the board.

Once you know the pin pitches and manufacturing requirements, you are ready to determine the routing grid for the autorouter. Using the LAYOUT tool, you can create four types of grids:

- **Uniform Grid**—a simple trace grid that provides equal grid spacing across the entire board. This type of grid is called a uniform grid. A uniform grid is usually spaced at .05, .025, .020, or .0125. Create a uniform grid by entering Trace Grid values in the Setup Routing Rules dialog box.

- **Non-uniform Grid**—direct LAYOUT to calculate a more complex, fine-line pin grid that provides more than one routing channel between padstacks. This type of grid is called a non-uniform grid. A non-uniform grid is based on the default padstack specifications for your design. Create a non-uniform grid by entering Pin Grid values in the Setup Routing Rules dialog box.

- **Combination Grid**—specify a combination uniform and non-uniform grid for boards with mixed pin pitches. A combination grid generally works well for a mix of surface and through-hole pin pads. Create a combination grid by entering both Trace Grid and Pin Grid values in the Setup Routing Rules dialog box.

- **Custom Grid**—define a uniform or pattern grid for all or just part of the board. You can merge the custom grid with an existing routing grid, or replace the current routing grid entirely. You can also add and delete individual grids anywhere on the board. Create a custom grid using the grid editor features.

Calculating the Number of Traces

The goal is to get as many traces between pads as possible. The method for determining if a grid is feasible is similar for both uniform and non-uniform grids. Use the following formula to determine how many traces you can theoretically get between each set of pads.

traces = (pad spacing - clearance - pad size) / (trace size + clearance)

The Through-hole pin Pad

First calculate the number of traces between pads for 54-mil diameter through-hole pin pads on 100-mil centers.

traces = (100 - 8 - 54) / (8 + 8) *or* **traces = 38 / 16**
traces = 2

A grid using these design rules allows two traces between the through-hole pin pads.

The Surface-mount Pad

Now calculate the number of traces between pads for 25-mil surface-mount (SMT) pads on 50-mil centers.

traces = (50 - 8 - 25) / (8 + 8) *or* **traces = 17 / 16**
traces = 1

A grid using these design rules allows one trace between the SMT pads.

Uniform Grid

The uniform grid is also referred to as the trace grid. In most SMT and MCM designs, the uniform grid gives best results, but the uniform grid is also useful for through-hole pin technology. You want to create a grid that yields the most traces between pads with no clearance violations.

The following rules and guidelines apply:

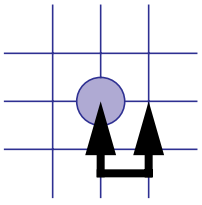
- When working in English units, it is best to use grid spacing that divides evenly into 100 mils (50, 25, 20, 12.5, or 10 mils). This is because most components have a pin pitch that is some multiple of 100 mils. This is a guideline, not a system requirement. For boards with components having a combination of pin pitches, a combination grid might provide better results.

Grids can also be calculated using metric units.

- When choosing a uniform grid, as the grid becomes finer the router needs more time to complete a pass and requires more computer memory. For example, a .010 inch grid is four times as complex as a .020 inch grid.

When calculating a uniform grid, be sure to take pad size, spacing between pads (pin pitch), trace width, and clearances into account. Remember that the auto router will not route a trace on a grid where a clearance violation would result. A simple formula for calculating the available clearance for a given grid spacing, pad size, and trace width is:

grid center - 1/2 pad diameter - 1/2 trace width = clearance



NOTE: ***Grid Center is the distance from a pin center to the first grid outside the pin pad.***

Using the Formula

Test a 25-mil grid using design characteristics as follows:

- Through-hole pin pads—54 mils diameter on 100-mil centers
- SMT pads—25 mils wide on 50-mil centers
- Trace width—8 mils
- Trace clearance—8 mils

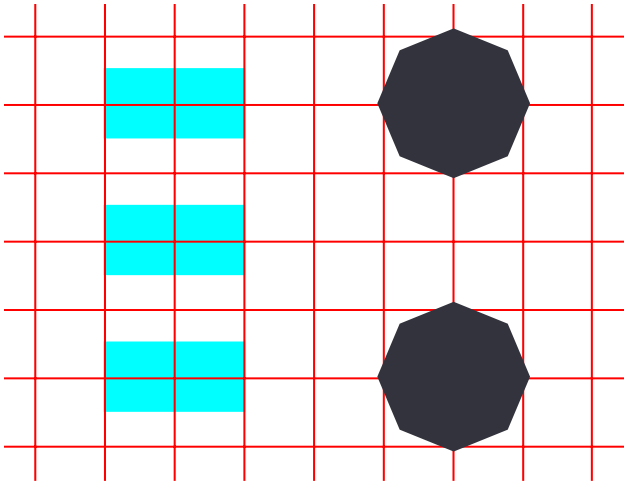


Figure 2-1. Example of a Uniform Grid

Use the formula to determine whether traces can route between the SMT pads using 25-mil grid spacing:
25 - 13 - 4 = 8

Do the same test for the through-hole pin pads:
50 - 27 - 4 = 19

Conclusion: the 25-mil grid is a good choice, because you can get traces between both types of pads.

Use **Setup Routing > Routing Rules...** to set:

- Trace Grid: x = 0.025, y = 0.025
- Pin Grid: x = 0, y = 0

Non-uniform Grid

The non-uniform grid, which is also referred to as the pin grid, has the advantage of reducing the total number of grids, while still providing a maximum number of grids between pads. A non-uniform grid is preferable to using a fine uniform grid. The fine uniform grid contains many extra, unneeded grid lines, resulting in slower routing time and a greater memory requirement. Using a non-uniform grid can provide a significant savings for large boards where a reduction in grids can result in faster routing time and a reduced memory requirement.

The reduced number of grids results from the calculation of a grid pattern based on:

- pin pitch.

  The spacing of the pins, or pin pitch, on the component type for which you are calculating. Pin pitch is what you specify in the Pin Grid entry box in the Setup Routing Rules dialog box.

- default trace width.

  The trace width specified in the net rule for the default net type.

- default trace clearance.

  The trace-to-trace clearance and pin-to-pin clearance specified in the net rule for the default net type.

- default pad size.

  The value of the Default_pad_size attribute defined in the board geometry.

LAYOUT automatically generates the pattern and locates a grid at every pin grid interval, then inserts as many additional grids between pin grid pairs as current trace width, trace clearance, and pad size allow.

Using the Formula

As an example, use the design characteristics as follows:

- Through-hole pin pads—54 mils diameter on 100-mil centers
- SMT pads—25 mils wide on 50-mil centers
- Trace width—8 mils
- Trace clearance—8 mils

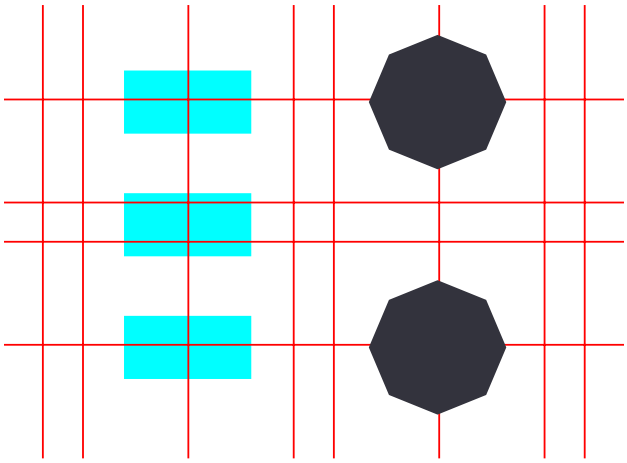


Figure 2-2. Example of a Non-Uniform Grid

First, calculate the number of traces between pads for the 25-mil SMT pads on 50-mil centers.

traces = (50 - 8 - 25) / (8 + 8) *or* **traces = 17 / 16**
traces = 1

A grid using these design rules allows one trace between the SMT pads.

Now, do the same test for the through-hole pin pads.

traces = (100 - 8 - 54) / (8 + 8) *or* **traces = 38 / 16**
traces = 2

A grid using these design rules allows two traces between the through-hole pin pads.

Setting the Pin Grid

When you enter Pin Grid values, you are actually specifying the pin pitch. LAYOUT automatically performs the actual grid calculation. Entering a trace grid of 0 tells LAYOUT not to generate a uniform grid because you are defining a custom grid.

Use **Setup Routing > Routing Rules...** to set:

- Trace Grid: x = 0, y = 0
- Pin Grid: x = 0.1, y = 0.1

Using the specified Pin Grid values, LAYOUT generates the pattern and locates a grid at every pin grid interval. LAYOUT then inserts as many additional grids between pin grid pairs as current trace width, trace clearance, and pad size allow.

Combination Grid

A combination grid overlays a non-uniform grid on a uniform grid. It can be useful in a design with mixed surface and through-hole pin padstacks of different pin pitches. Its advantage is an ability to generate grids between both surface and through-hole pin padstacks when neither the uniform nor non-uniform grid alone could achieve the desired pattern.

Using the same design characteristics as in previous examples:

- Through-hole pin pads—54 mils diameter on 100-mil centers
- SMT pads—25 mils wide on 50-mil centers
- Trace width—8 mils
- Trace clearance—8 mils

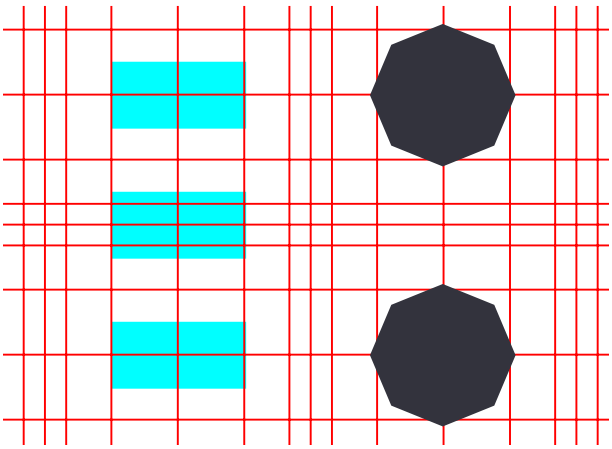


Figure 2-3. Example of a Combination Grid

Use **Setup Routing > Routing Rules...** to enter trace and pin grid values. Entering values for both Trace and Pin grids creates both uniform and non-uniform grids.

- Trace Grid: x = 0.025, y = 0.025—as the uniform grid.
- Pin Grid: x = 0.1, y = 0.1—as the pin pitch for LAYOUT to use in automatically calculating the non-uniform grid.

Custom Grid

The LAYOUT custom grid features allow you to design and save a routing grid that is separate from the design grid. You can merge a custom grid with a design grid or replace the design grid for routing your board.

The design grid is the routing grid set in the Setup Routing Rules dialog box, or specified with any of the optional board grid attributes: Board_wire_grid, Board_pin_grid, or Board_via_grid.

Custom grid features include the following:

- Create uniform or pattern grid
- Delete grid
- Add/Delete individual grids interactively

Custom grid features are best for making localized changes to a routing grid. For example, you see an area where no grid exists and the addition of a grid line would aid autorouting, or you want to remove grid lines where they are not helpful. Removing unnecessary grid lines can speed up autorouting.

LAYOUT saves the custom grid by default as the *grid* design object. Each time you invoke LAYOUT on the design, LAYOUT restores the *grid* design object as the routing grid for the session. Subsequent changes to the trace, pin, and via grid settings in the Setup Routing Rules dialog box do not override a custom grid unless you choose to do so when starting the autorouter.

Interactive Grid Editor

You can create a custom grid by completing the Change Routing Grid dialog box:

Auto Routing > Change Routing Grid

or by using the interactive grid editing features.

Auto Routing > Change Routing Grid > Add Routing Grid Line:

Auto Routing > Change Routing Grid > Delete Routing Grids:

You can add an individual grid at any location on the board along either or both axes. You can delete all grids within an area along either or both axes. Grids are added and deleted along the chosen axes to the full extent of the routing area of the board. In other words, if you add a grid at a point along the x axis, it extends vertically above and below the point to intersect the routing outline. If you delete grids along the y axis within an area, all horizontal grids within the area are deleted from one side of the routing outline to the opposite side.

Change Routing Grid Dialog Box

To produce a customized routing grid, select the **Auto Routing > Change Routing Grid** menu item and complete the dialog box.

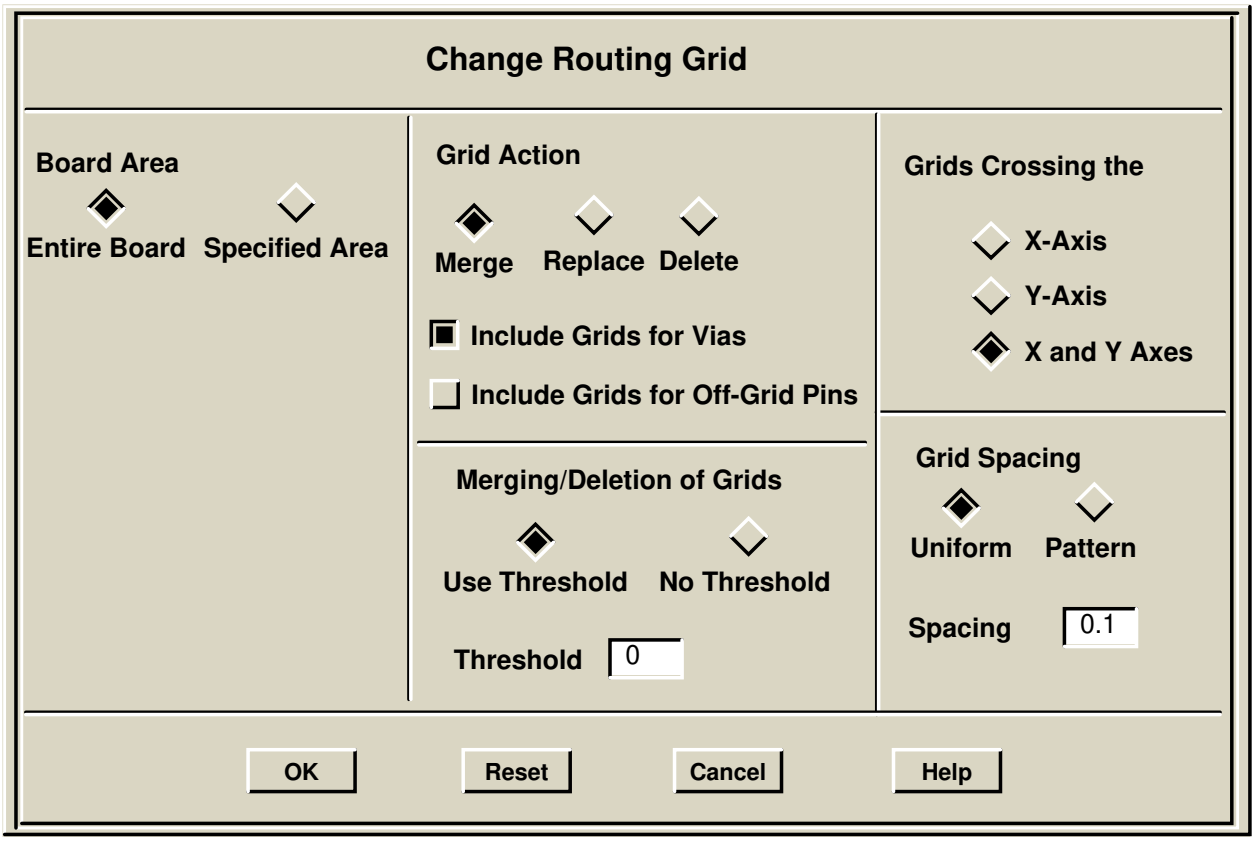


Figure 2-4. Change Routing Grid Dialog Box

- **Board Area**—applies the Grid Action and other choices to the entire board or to a specified area of the board. If you choose to specify an area of the board, you can specify a location for the origin of the new grid.

- **Grid Action**—specifies how to implement the custom grid. Choices allow merging the specified grid with the existing grid, replacing the existing grid entirely, or deleting the existing grid.

If you choose Delete as the grid action, you are only deleting the specific grid that you defined using the controls in the dialog box. For example, if you previously defined a combination grid, then select to delete a uniform grid, you delete only the uniform portion of that combination grid.

- **Merging/Deletion of Grids**—provides a choice of threshold distance when merging or deleting grids. An existing grid that lies closer to a new grid than the threshold distance is deleted. This choice prevents the creation of too many grid lines that slow the autorouter.
- **Grids Crossing the**—applies the Grid Action choice to the X or Y axis, or both.
- **Grid Spacing**—sets the grid interval(s) for the Grid Action. You can specify a uniform grid or a pattern. Using pattern spacing, you can create the same kind of grid spacing LAYOUT generates when you specify a non-uniform grid or a combination grid.

Grid Spacing

When you set grid spacing in the Change Routing Grid dialog box, it is important to understand how the spacing interval you enter affects the spacing of the grid lines in both the X and Y directions.

Spacing between grids crossing the X-axis, is actually Y direction spacing. Spacing between grids crossing the Y-axis, is actually X direction spacing.

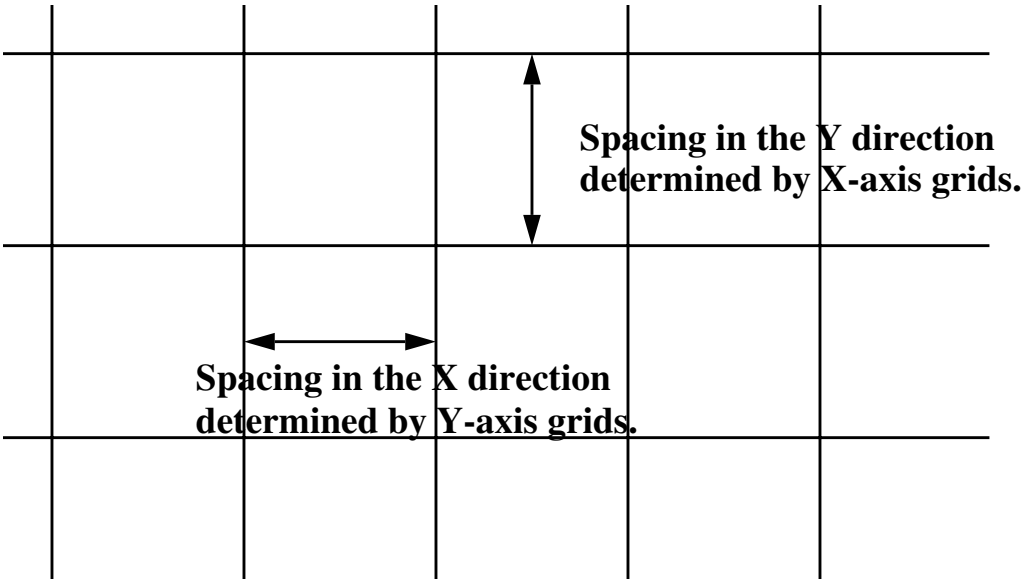


Figure 2-5. Spacing Grid Lines

Auto Router Operation

The auto router operates with design rules specified for nets and layers, and according to the current routing rules. These sets of design and routing rules affect both interactive and automatic routing.

To begin autorouting, you select the routing algorithms you want to run and the number of passes you want the auto router to execute for each. You also specify other parameters, such as whether to route the entire board, to route just within an area, or to route only selected nets. You also decide whether to use the design grid instead of a custom grid, and how to save the routing.

Automatic routing progresses using the algorithms in the sequence you specified. The auto router decides which cost schedules to use based on design and routing rules, autorouting parameters you set, and percentage of completion after each pass.

Auto Router Setup

You can use the Setup Routing Rules dialog box to set up grids, as well as set additional constraints to control the autorouter.

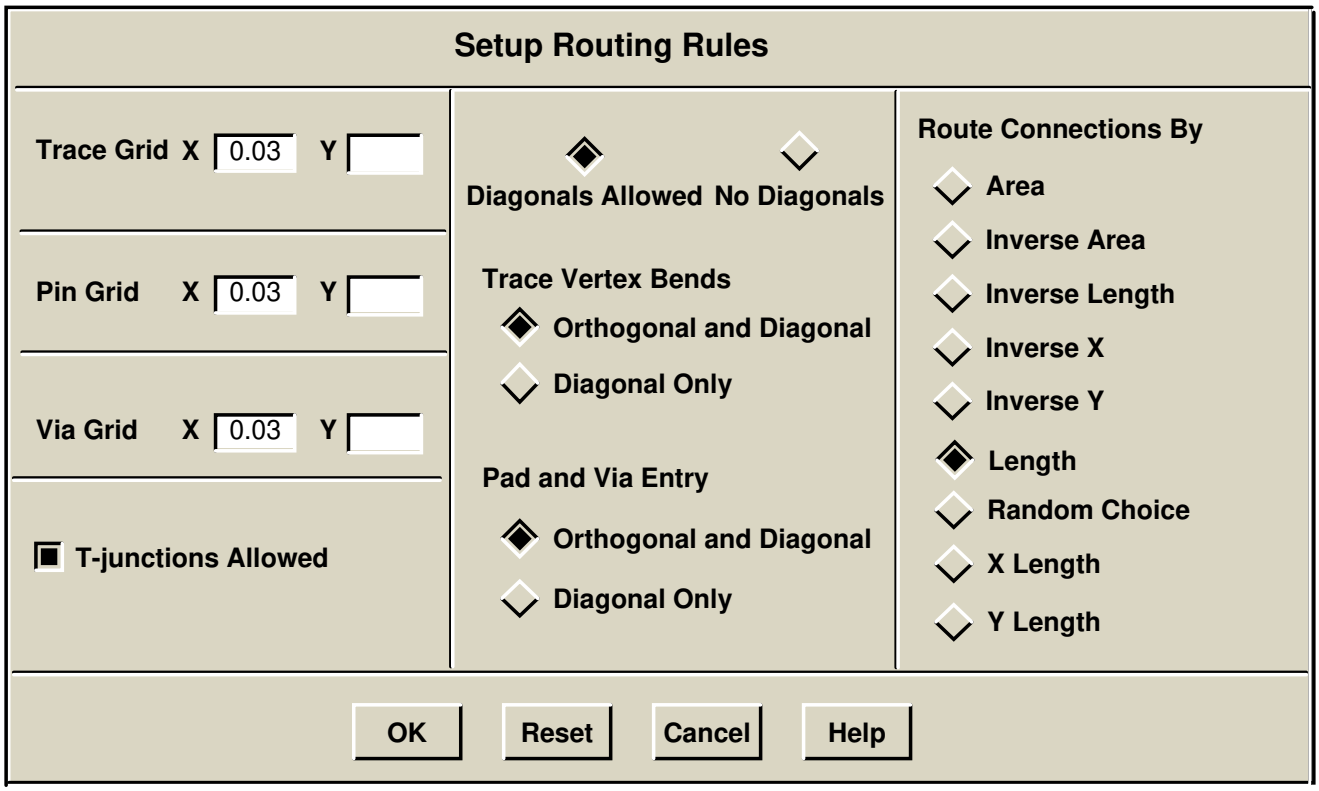


Figure 2-6. Setup Routing Rules Dialog Box

This opens the Setup Routing Rules dialog box with the following choices:

- **Trace Grid**—provides entry boxes for X and Y values that define a uniform grid. If used in combination with **Pin Grid** values, a combination grid results.
- **Via Grid**—provides entry boxes for X and Y values that define a via grid.

- **Pin Grid**—provides entry boxes for X and Y values that define a non-uniform grid. If used in combination with **Trace Grid** values, a combination grid results.
- **T-junctions Allowed**—specifies whether t-junctions are allowed. This overrides the optional attribute, Tjunctions_allowed, assigned to the board geometry in LIBRARIAN.
- **No Diagonals**—disables use of diagonal routing.
- **Diagonals Allowed**—enables diagonal routing and displays additional choices for limiting diagonal bends and pin entry.
- **Route Connections By**—specifies the sorting order of guides for automatic routing. Specifying to route by Length means the shortest guides route first. Specifying to route by Inverse X or Inverse Y means the longest guides route first.

Routing rules are saved with the environment in the <design>/*startup/layout.env* file.

Connection Sorting

Prior to the routing process, the automatic router sorts all connections to be routed. Nine different sorting orders are available. The different sorting orders allow you to prioritize routing connections based on the characteristics of your design.

	X	Y	AREA X * Y	LENGTH X + Y
A	6	5	30	11
B	1	18	18	19

* Sorting by AREA - guide to B has priority

* Sorting by LENGTH - guide to A has priority

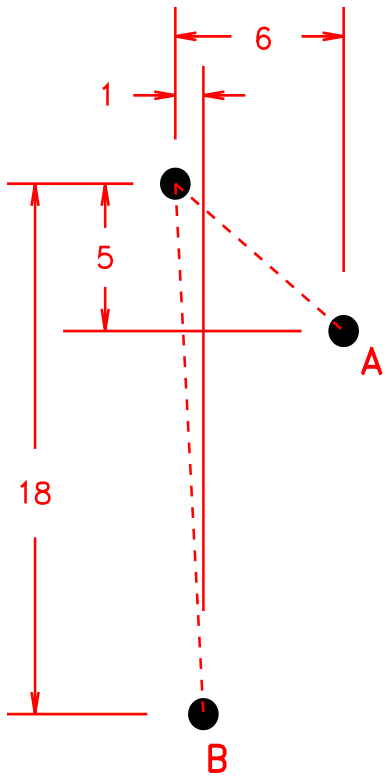
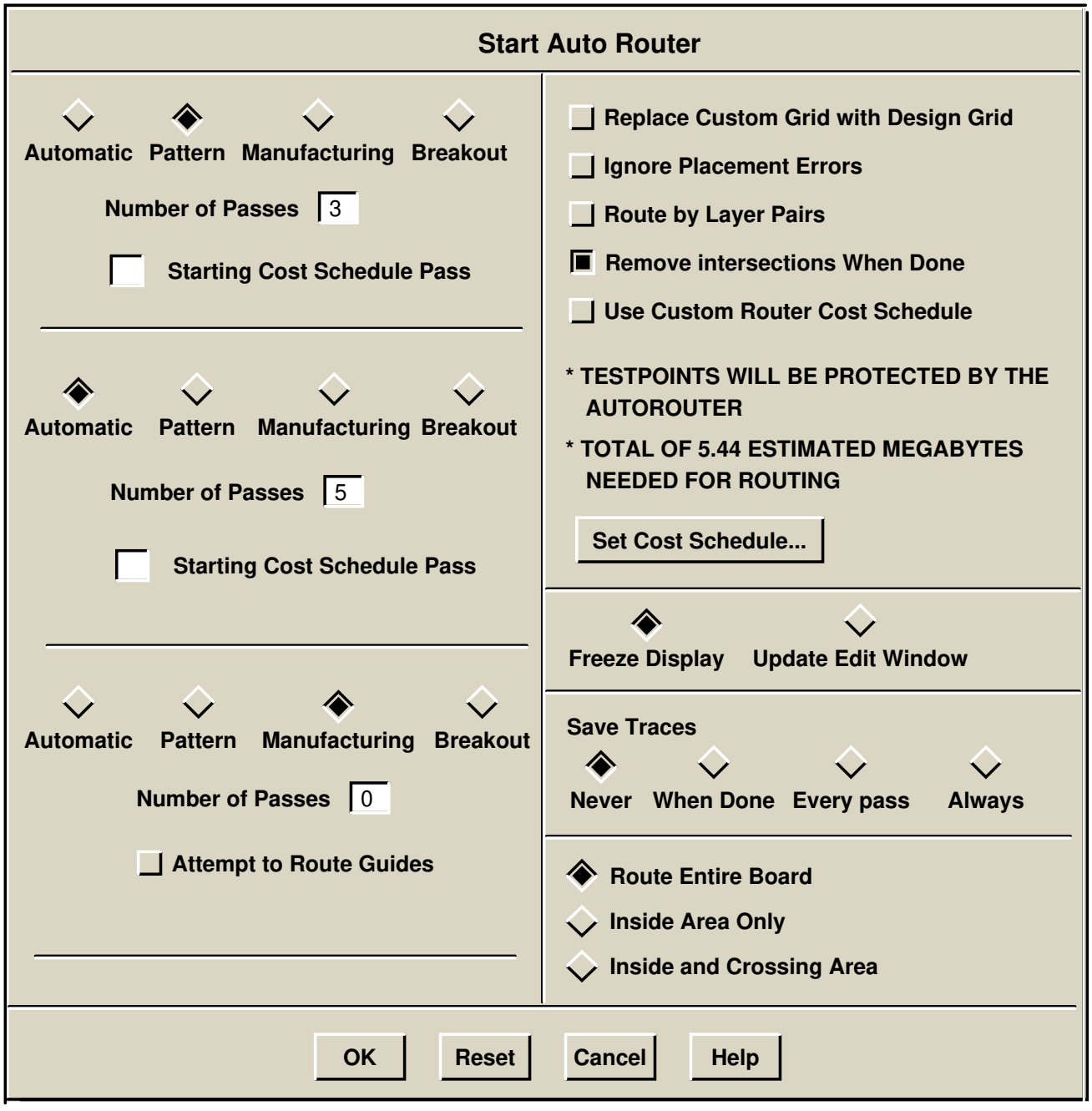


Figure 2-7. Sorting Connections to Determine Routing Order

Auto Router Theory



Algorithms

- **Pattern**—an additive router that routes busses and inserts power vias. It is useful when routing boards containing large bus structures.
- **Automatic**—a general purpose rip-up and re-try router that uses squeeze-through and shove-aside techniques to converge on an optimal routing solution through a number of passes.
- **Manufacturing**—a cleanup router that re-routes to eliminate unnecessary bends and reduce the number of vias. The manufacturing algorithm tends to remove vias and close channels.
- **Breakout**—a special case of the automatic router, which identifies eligible SMT padstacks, then routes a breakout pattern at the padstack. The breakout algorithm pulls a trace out to the first available via location and leaves the guide hanging.

Multiple Passes

You can specify any one or a combination of these algorithms to run in sequence for a number of passes. Multiple passes allows the router to converge to the highest possible completion percentage.

Routing Costs

How the auto router chooses one possible path over another is based on predefined costs associated with each possible choice. As the router works to complete a trace from one pin to another, it adds up the costs of the decisions made at each point in the attempted path. If the total cost for the path is higher than allowed, the trace is ripped up and tried again on the next pass. The costs applied during each pass vary according to a schedule you predefine that takes into account such factors as percentage of routing completion and specified routing parameters. The predefined cost schedules have been carefully designed to provide the best possible results for a wide variety of designs.

Lab Exercise

In this lab exercise you route traces on the circuit board using the automatic router. You first choose the routing options. Then, you route the design. You route several single nets, some nets associated with specific components, and finally, all remaining nets. Try your best strategies as you route the remaining nets. Your design must be completely routed at the end of this lab exercise.

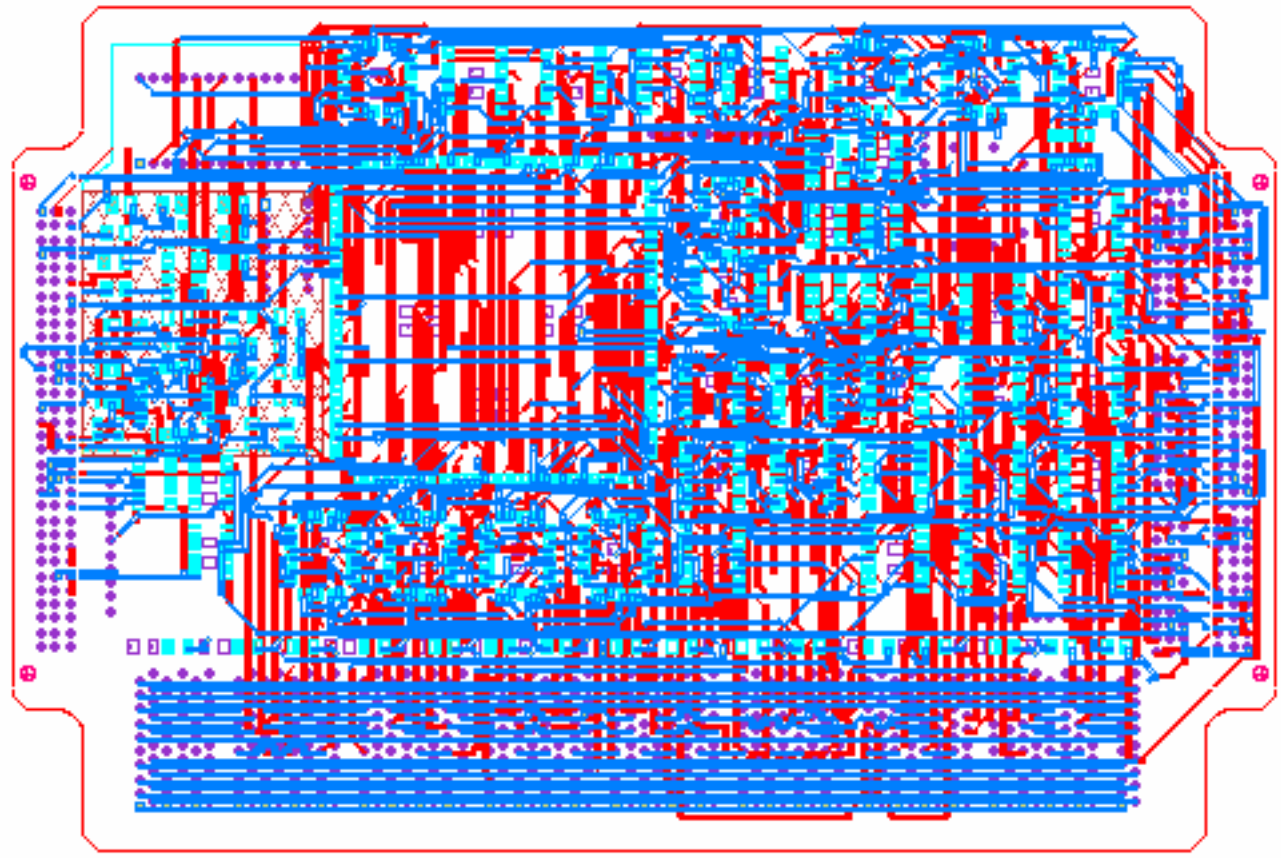
Upon completion of this lab exercise you are able to:

- Set up routing parameters.
- Select and route specific nets.
- Route nets from selected components.
- Route the entire design.

Turn to Module 6—Lab 2: "Automatic Routing".

Lab 2
Automatic Routing

This lab exercise familiarizes you with routing a board using the automatic routing tools.



Introduction

In this lab exercise, you route traces on the circuit board using the automatic router. First, you choose the routing options, route the design, route several single nets, nets associated with specific components, and finally, all remaining nets. Your design is completely routed at the end of this lab exercise.

Upon completion of this lab exercise, you are able to:

- Set up routing parameters.
- Define the routing grid.
- Select and route specific nets.
- Route nets from selected components.
- Route the entire design.

Procedure

In this lab exercise you use the LAYOUT tool to automatically route the remaining traces on the board.

Preparation for Lab

1. Invoke the Design Manager. Next, invoke LAYOUT on the **sig_az** design using the LAYOUT icon in the Tools window.

2. Close the report window.

Setting up Routing Grids

To define a combination grid for autorouting the design, you create a uniform trace grid in both X and Y directions. The pin grid values direct LAYOUT to create a non-uniform grid based on the pin grid values, as well as the default line width and clearances. Together, the uniform and non-uniform grids create a combination grid.

1. Choose the **Setup Routing > Routing Rules...** menu item. In the dialog box, enter the following, and then **OK** the dialog box.

This is the default autorouting grid.

Trace Grid X: **0.025** Y: **0.025**

Pin Grid X: **0.1** Y: **0.1**

Via Grid X: **0.05** Y: **0.05**

T-junctions Allowed

Diagonals Allowed

Trace Vertex Bends: **Orthogonal and Diagonal**

Pad and Via Entry**: Orthogonal and Diagonal**

Route Connections By: **Length**

The default net rules are always used in the grid definition, even if you specify another net rule. Next you define the default net rules.

2. Choose the **Setup Routing > Net Rules...** menu item. In the dialog box, choose **DEFAULT_NET_TYPE**, then select **Change...** In the Change Net Rules dialog box, ensure the net rules are set as follows. Then **OK** the dialog box.

   Trace Width: **0.008**

   Pin to Pin Clearance: **0.008**

   Via to Via Clearance: **0.008**

   Pin to Trace Clearance: **0.008**

   Via to Trace Clearance: **0.008**

   Pin to Via Clearance: **0.008**

   Trace to Trace Clearance: **0.008**

   Select Interactive Routing Vias: **via040015**
   via_thru

   Routing Vias for Auto Routing: **via040015**
   via_thru

To select a via for autorouting, it must also be selected for interactive routing.

Available Routing Layers: **Trace_Layer_1**
Trace_Layer_2
Trace_Layer_3
Trace_Layer_4

To select more than one non-sequential item from a list, hold down the CTRL key and then click the Select mouse button on the second and subsequent selections.

3. Set the same net rules for net types 50_OHM and DIFF_PAIR.

4. Ensure these same net rules are set for the POWER_NETS and POWER_15 net types with the following exceptions:

   Trace Width: **0.025**

   Select Interactive Routing Vias: **power_via_front**
   power_via_back

   Routing Vias for Auto Routing: **power_via_front**
   power_via_back

5. **Close** the Setup Net Rules dialog box.

   Now that you have defined the routing grid, you can generate and display it.

6. Choose the **View > View Routing Grid** menu item. In the prompt bar, click on the up/down scroll arrows until **entire_board** is displayed in the prompt bar, then **OK** the prompt bar.

   The grid takes a minute to generate. After LAYOUT completes the grid generation, a Report-Display Grid message appears in the middle of the LAYOUT Session window.

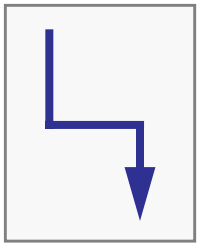
When changing route grids, you must refresh the display to view the changes.

7. Read the report and look for the heading "Memory Usage Report". Notice the amount of memory required for the grid. After reading the report, close the report window.

Experimenting with Routing Grids

1. Change the view area to a small section of the board so you can see the grid better. It does not matter what area you view.

   Table 2-1 defines other sample grid options and indicates the grid type. The via sites are displayed as diamonds at grid crossings. The system places a via only at the indicated locations during automatic routing.



2. Display the Display Controls dialog box. Turn off the **Guides** object temporarily. In the Interactive Routing Aids section, note that the **Route Grid** object is selected.

   Generating a routing grid automatically selects the Route Grid object. With the grid generated, you can control visibility by selecting or deselecting the Route Grid object. You can interpret the grid using the following guidelines.

   - Diamonds are potential via sites.
   - Plus signs indicate available routing channels based on the potential via sites.
   - X signs indicate blocked routing channels.

   If you want to experiment with different grids, you can change the routing rules and net rules to correspond to some of the Grid Setup Options suggested in Table 2-1. You can then view the routing grid you create to determine which option creates a grid with the best coverage of routing channels and the least number of off-grid pins.

   Before you begin automatic routing, delete the grid and create a specific grid for the lab design.

Table 2-1. Grid Setup Options

	Uniform		Non-Uniform			Combination	
	A	B	C*	D*	E	F	G
Trace Width	0.012	0.012	0.012	0.008	0.008	0.008	0.008
Trace Clearance	0.013	0.013	0.013	0.008	0.008	0.008	0.008
Trace Grid	0.025	0.025	0.0	0.0	0.0	0.025	0.025
Pin Grid	0.0	0.0	0.0	0.0	0.1	0.1	0.1
Via Grid	0.0	0.05	0.05	0.05	0.0	0.0	0.05

* This is not a true non-uniform grid, even though it displays like one. A non-uniform grid requires a pin grid rule.

Creating Custom Routing Grids

If that the system-defined grid does not meet your layout requirement, you can customize the system-defined routing grid or create a completely customized grid.

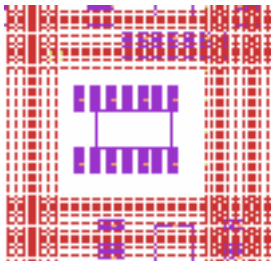
In the next steps, you learn how to change grids in specific areas of the board. Do not be concerned with creating a final grid as you create a new one just before autorouting the board.

1. Use the View Area stroke to view a small area of the board (about the size of a component), so you can see the individual grid lines.

2. Choose **[Auto Routing] Change Routing Grid > Delete Routing Grids:** to display the prompt bar.

3. In the Along Axis prompt, choose **xy** and **norepeat**. Tab to the location prompt, or click on it with the Select mouse button.

   Place the cursor on a grid point, hold down the Select mouse button, and drag the cursor so a select area box encloses all the grid lines you wanted to remove. Release the Select mouse button.



   The X and Y grid lines are removed. Now, add a grid line.

4. Choose the **[Auto Routing] Change Routing Grid > Add Routing Grid Line:** menu item. In the prompt bar, choose **Options**, and complete the dialog box as shown. **OK** the dialog box.

   Along XY Axis
   Include Grid for Via
   [Unselect the repeat option]

5. When LAYOUT prompts for a location, place the cursor on a grid point that does not have a grid line running through it (such as the area in which you deleted grid lines), and click the Select mouse button.

   Grid lines are created in the X and Y directions.

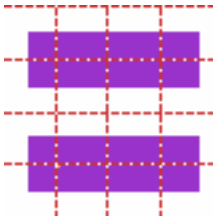
Creating a Uniform Grid

To clearly see the difference in the autorouting grids, you create a new uniform grid. Again, you completely replace the existing grid.

1. Choose the **[Auto Routing] Change Routing Grid > Change Routing Grid...** menu item. In the dialog box, enter the following, and then **OK** the dialog box.

   Board Area: **Entire Board**

   Grid Action: **Replace**



   Include Grids for Vias

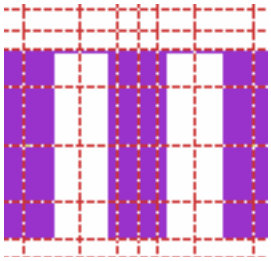
   Grids Crossing the: **X and Y Axes**

   Grid Spacing: **Uniform**

   Spacing: **.025**

2. Refresh the view to see the changes. View areas around the board that have various types of components. Note how appropriate or inappropriate this grid is for the various components.

Creating Non-uniform Grid Pattern



Use the Change Routing Grid dialog box to create a new complex grid over the entire board. You completely replace the existing grid with the one you define using the controls in the dialog box.

1. Choose the **[Auto Routing] Change Routing Grid > Change Routing Grid...** menu item. Enter the following in the dialog box, and then **OK** the dialog box.

   Board Area: **Entire Board**

   Grid Action: **Replace**

   Include Grids for Vias

   Grids Crossing the: **X and Y Axes**

   Grid Spacing: **Pattern**

   Pattern: **.025 .016 .009 .009 .016 .025**

   The new complex grid is created.

2. Refresh the view to see the changes. View a small area of the board, so you can see the grid lines clearly and see the result of the new grid specifications.

This grid contains lines through the surface mount pads that were off grid using the uniform grid method. However, this grid does contain many extra grid lines that are not needed and can slow routing. Try an alternate method of creating grid lines for the surface mount pads.

Creating a New Uniform Grid

To clearly see the difference in the autorouting grids, you now create a new uniform grid. Again, you completely replace the existing grid.

1. Choose the **[Auto Routing] Change Routing Grid > Change Routing Grid...** menu item. In the dialog box, enter the following and then **OK** the dialog box.

   Board Area: **Entire Board**

   Grid Action: **Replace**

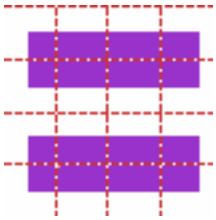
   Include Grids for Vias

   Include Grids for Off Grid Pins

   Grids Crossing the: **X and Y Axes**

   Grid Spacing: **Uniform**

   Spacing: **.025**



2. Refresh the view to see the changes. View areas around the board. Note that there are fewer grid lines than with the non-uniform grid, and that the surface mount pads also have route grid lines through them.

Reset the Routing Grid

Next, reset the routing grid to the design default. You use this grid for autorouting.

1. Choose the **[Auto Routing] Change Routing Grid > Reset to Design Rules...** menu item. In the Reset Routing Grid dialog box, choose **OK**. Refresh the view to see the changes.

2. Choose menu item **View > Display Controls**. In the Interactive Routing Aids section, turn off **Route Grid**.

Setting Up for Autorouting

Before starting the autorouter for the first time, it is good design practice to check the routing rules. In addition, verify the net rules are correctly defined. After you set the rules, they are saved with the design.

1. Choose the **Setup Routing > Routing Rules...** menu item. Verify the routing rules are set as follows, then **OK** the dialog box.

Trace Grid X: **0.025** Y: **0.025**

Pin Grid X: **0.1** Y: **0.1**

Via Grid X: **0.05** Y: **0.05**

T-junctions Allowed

Diagonals Allowed

Trace Vertex Bends: **Orthogonal and Diagonal**

Pad and Via Entry**: Orthogonal and Diagonal**

Route Connections By: **Length**

2. Choose the **Setup Routing > Net Rules...** menu item. In the dialog box, choose **DEFAULT_NET_TYPE**, then select **Change...**

In the Change Net Rules dialog box, verify the default net rules are set as follows, then **OK** the dialog box.

Trace Width: **0.008**

Pin to Pin Clearance: **0.008**

Via to Via Clearance: **0.008**

Pin to Trace Clearance: **0.008**

Via to Trace Clearance: **0.008**

Pin to Via Clearance: **0.008**

Trace to Trace Clearance: **0.008**

Select Interactive Routing Vias: **[Select all vias that appear]**

Choose one or more of the SELECTED Interactive Routing

Vias for AutoRouting: **via040015**
via_thru
Available Routing Layers: **Trace_layer_1**
Trace_layer_2
Trace_layer_3
Trace-layer_4

3. **Close** the Net Rules dialog box.

4. Choose menu item **Setup Routing > Physical Layers > Via Rules**.

5. In the Change Via Rules dialog box, select **via040015** and then select the **Setup Rules** button. In the Rules for Via Under Pad section, select **Trace_Layer_1**, and then select **Connect**.

6. In the Connect Via Rules dialog box, indicate that Connections to Single Layer Pin Pads are **Not Allowed**. **OK** the Connect Via Rules dialog box.

7. **Close** the Setup Buried or Two-layer Via Rules dialog box.

Saving Existing Traces

In this section, you route all traces on the board using the auto router. First, save the existing traces so you can restore them later.

1. Choose the **File > Save > to Design Object...** menu item. In the Save to Design Object dialog box, select **Traces**, enter the name **traces_saved_1**, and then **OK** the dialog box. In the Save Traces Options dialog box, choose the following, and then **OK** the dialog box.

   Save Guides
   Save Partial Routes

   You can use this method to save several variations of the traces. Because you must specify a design object name, in this case *traces_saved_1*, you can save many design objects with trace information.

   After saving the trace information, unprotect the traces and then select and delete them.

2. From the Auto Route palette, choose the **Unprotect Routing** icon.

3. In the Unprotect Routing dialog box, select **All Traces** and **OK** the dialog box.

4. The traces return to their normal color, indicating they are not protected. You can select and delete them as follows.

5. From the Edit window, choose the **[Auto Routing] Select > Select All Traces** menu item.

6. Choose the **[Auto Routing] Delete...** menu item. In the dialog box, choose **Traces**, and then **OK** the dialog box.

   All traces are removed.

Running Breakout Pass

You are ready to start the autorouter. Because there are surface mount components on the board, you might want to run a breakout pass first. The breakout pass *breaks out* traces away from surface mount pads to vias.

1. View all the board, so you can see the traces that are routed.

2. From the Auto Route palette menu, choose the **Auto Route** icon. Complete the Start Auto Router dialog box as shown, and then **OK** the dialog box.

   Breakout
   Number of Passes: **1**
   Breakout options: **No**

   Automatic [Leave the remainder of this section blank]

   Remove Intersections When Done

   Update Edit Window
   Trace Changes per Update: **100**

   Save Traces: **Never**

   Route Entire Board

   The route takes a few minutes to run. As the system is routing, a Report-Auto Router message appears in the middle of the LAYOUT Session window.

3. Close the report Notepad when the route is complete.

Predicting Routability

Now that you have routed the breakouts, use the Routability Predictor to evaluate the routability.

1. Choose the **Report > Routability** menu item. **OK** the dialog box with the default information.

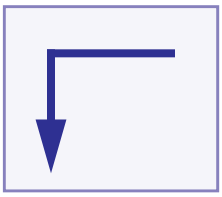
   This function takes a few minutes to process the board placement data. You can monitor the progress by watching the status window. The result of this function is a routing density display that indicates the locations of difficult to route areas. 100 percent indicates that all of the available routing channels are filled. Any 100 percent or 90 percent areas indicated on the board are be more difficult to route than other areas. If you have any areas that are indicated to be 100 percent or 90 percent filled, consider moving components to provide more space for routing.

   When the report is complete, the report window displays.

2. Close the report window. Use the View All stroke to see the scale range on the left side of the board.

3. After looking over the scale range, clear the report by choosing the **Report > Routability > Clear Report** menu item.

Protect the Routing



To keep the autorouter from ripping up and rerouting your breakouts, protect the traces before running the autorouter again.

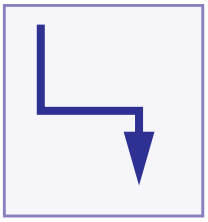
1. Use the Setup Select Filter stroke and check the Select Filter to ensure that **Traces**, **Vertices**, and **Vias** can be selected.
2. Choose menu item **[Auto Routing] Select > Select All**.

3. From the palette, choose the **Protect Routing** icon.
4. In the Protect Routing dialog box, ensure **Protect Selected Routing** is selected. Protect Groups must be set to No and Protect Nets to None.
5. **OK** the dialog box.

Routing the Connector

1. Choose the **[Auto Routing] Unselect** menu item.

   It is okay if a menu item is grayed out, as you are just verifying that nothing is selected.



   In the next step, you select only nets connected to the connector for routing.

2. Use the Display Controls stroke, select the Guide object, and **OK** the dialog box.
3. Choose the **[Auto Routing] Select > Select Area:** menu item. From the prompt bar, click the Options button, and choose **Nets** from the Options dialog box. **OK** the dialog box. Use the Select mouse button to select an area around the J1 connector on the left side of the board; this selects all the nets that connect to the connector.

   Route the selected nets.

If nets are selected, the autorouter only routes those nets. If nothing is selected, the autorouter routes all nets.

4. Choose the **[Auto Routing] Start Auto Router...** menu item. In the dialog box, enter the following, and then **OK** the dialog box.

Pattern
Number of Passes: **2**
Starting Cost Schedule Pass: **[Leave this box at default]**

Automatic
Number of Passes: **5**
Starting Cost Schedule Pass: **[Leave this box blank]**

Automatic [Leave the remainder of this section blank]

Remove Intersection When Done

Update Edit Window
Trace Changes per Update: **100**

Save Traces: **Never**

Route Entire Board

Only the selected nets are routed.

This time the router takes a little longer, because it is making more passes.

5. Close the report Notepad when the route is complete.

Route the Remainder of the Board

1. Choose the **[Auto Routing] Unselect** menu item.
2. View the entire board, so you can see all the routed traces.

3. From the Auto Route palette menu, choose the **Auto Route** icon In the dialog box, enter the following, and then **OK** the dialog box.

   Pattern

   Number of Passes: **3**

   Starting Cost Schedule Pass: **[Leave this box at default]**

   Automatic

   Number of Passes**: 7**

   Starting Cost Schedule Pass: **[Leave this box blank]**

   Manufacturing

   Number of Passes**: 2**

   Attempt to Route Guides

   Automatic [Leave the remainder of this section blank]

   Remove Intersections When Done

   Update Edit Window

   Trace Changes per Update: 100

   Save Traces: Never

   Route Entire Board

   This routing takes about an hour.

4. Close the report Notepad when the route is complete.

   You might notice that some nets have not been routed. Whether the board routes to 100% or not depends on the exact placement of the components. In the next lab session, all unrouted nets are routed.

Tracking a Net

Applying only a pattern to a net allows you to track the net through the layers of the board. The net retains the color assigned to each layer, yet highlights in the unique pattern.

1. Select **View > Set Display Attributes** to display the dialog box. This is the same dialog box that you can access through the Display Controls dialog box.

   Note the many patterns and colors available and the objects and layers to which you can assign attributes.

2. Click on the **Net Highlight** button to see the list of nets.

   You can assign display attributes to nets. When you highlight an individual net, the net highlights using the color and pattern you set.

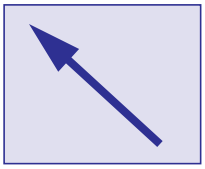
3. Select net **/RW**. Apply a unique pattern to the net.

4. **OK** the dialog box.

   Now highlight net /RW.

5. Select menu item **[Routing] > Highlight Net**.

6. In the prompt bar, enter net name **/RW**. Use the stepper button to set the **norepeat** option and **OK** the prompt bar.



7. Use the View All stroke to view the entire board.

8. Select the **View > Layers** menu item. Turn off visibility for layers Signal, Signal_1, Signal_2, Signal_3, and Signal_4. **OK** the dialog box.

   The highlighted net is still visible even though the layers on which the net resides are not set to visible.

9. Zoom in on part of the highlighted net closely enough to see the color and pattern of the net.

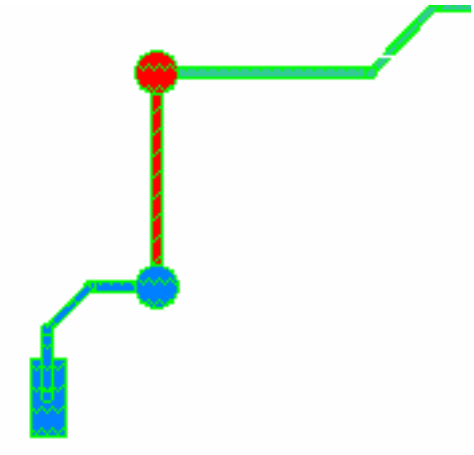


Figure 2-8. Tracking a Net with Pattern Highlighting

Because you assigned a pattern only to the net, you can track the net from layer to layer. The net retains the color assigned to the layer and is identifiable by the highlight pattern.

10. Set layers Signal, Signal_1, Signal_2, Signal_3, and Signal_4 to visible again.

11. Highlight Net is a toggle. To unhighlight the /RW net, select the **Highlight Net** menu item.

12. Enter **/RW** in the typing box and set the stepper button to **norepeat** in the prompt bar. **OK** the prompt bar.

    Net /RW no longer highlights.

Viewing Edit Layers

1. With the signal layers visible, make sure you zoom in far enough to clearly see the traces on top of each other.

   Since objects are opaque, you can see which layer is on the top. The top layer is the Edit layer.

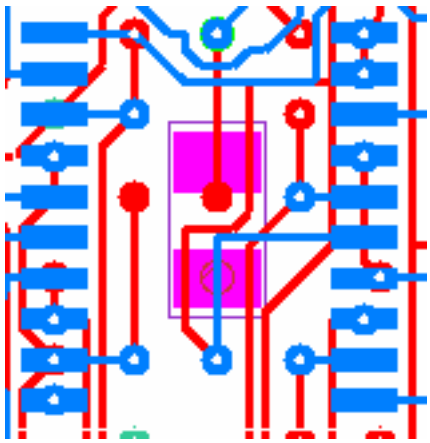


Figure 2-9. Edit Layer Appears on Top

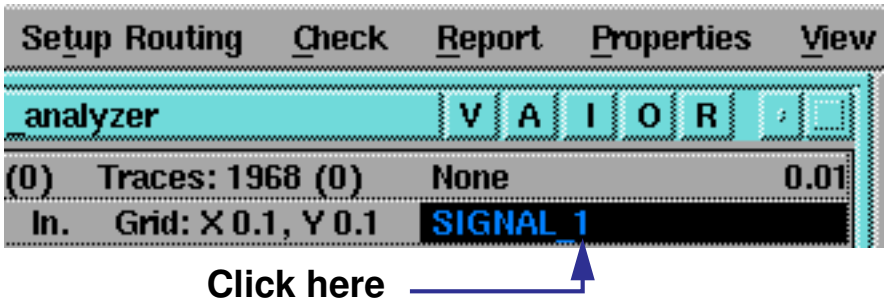


Figure 2-10. Edit Layer Display Area

2. Click on the Edit layer display area and cycle through the Edit layers.

Note that each time you change the Edit layer, the objects on that layer pop to the top.

3. Select **View > Display Controls** to display the dialog box. Select to view objects on the **Edit Layer Only**.

4. Click on the Edit layer display area and cycle through the Edit layers. Note that only objects on the Edit layer display. Objects on other layers do not display.

5. In the Display Controls dialog box, turn off **Edit Layer Only.**

6. Select the **Setup Routing > Edit Layer** menu item.

7. In the Set Edit Layer dialog box, set only layers Signal_1 and Signal_2 as edit layers.

8. Ensure the Edit layer is set to Signal_1. **OK** the dialog box.

Figure 2-11. Signal_1 and Signal_2 Edit Layer Set

This leaves an Edit layer set of Signal_1 and Signal_2.

9. Now click on the Edit layer display area and see how it only cycles through the current Edit layer set.

Experimenting with Routing

If time permits, change the following elements in any combination.

- Placement
- Routing rules
- Routing grids

Reroute the board. It is not important that you end up with a completely routed design. It is important that you become familiar with the elements that affect autorouting.

Saving the Routing

When you finish routing, save the traces so you can restore them later, if needed.

1. Choose the **File > Save > to Design Object...** menu item. In the Save to Design Object dialog box, select **Traces**, and enter the name **traces_saved_2**. Then **OK** the dialog box. In the Save Traces Options dialog box, choose the following, and then **OK** the dialog box.

   Save Guides
   Save Partial Routes

2. Choose the **Setup Routing > Routing Rules** menu item. In the dialog box, leave all the grid settings as they are, and choose **No Diagonals**, then **OK** the dialog box.

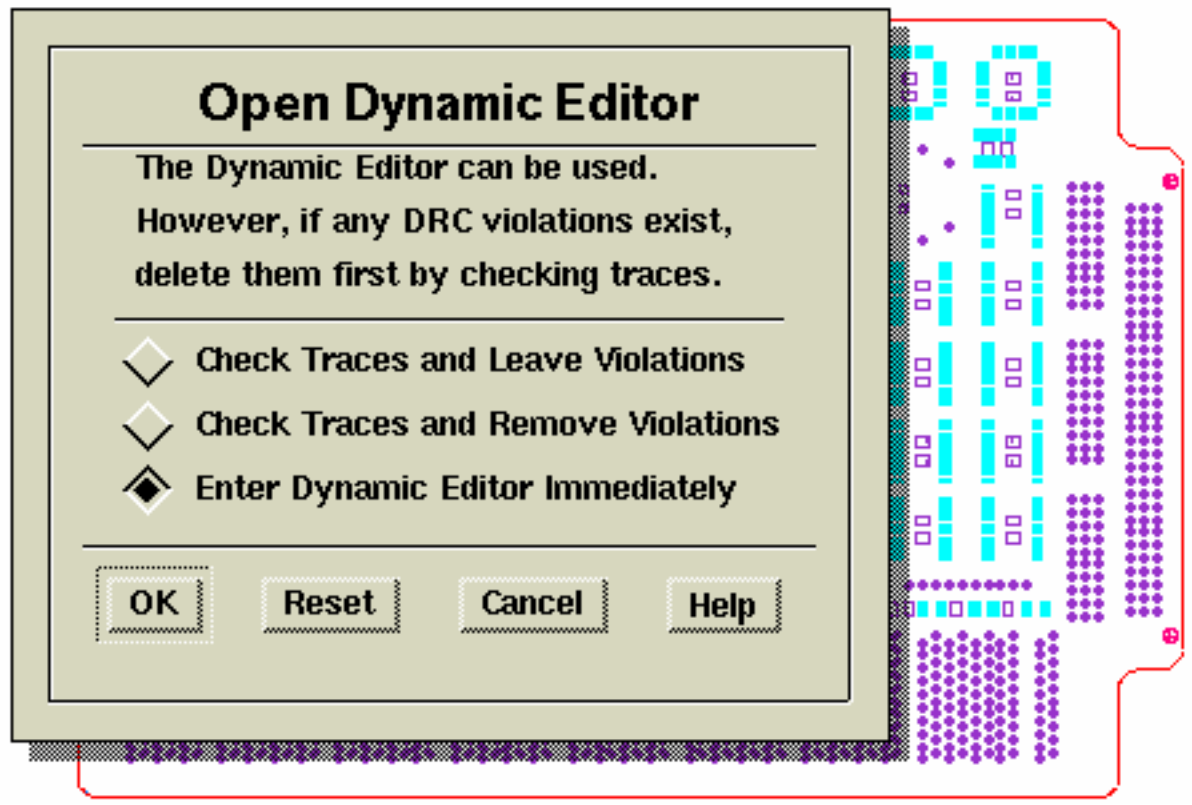
   You see the effect of this design rule change in the next lab exercise.

3. Choose **File > Save > Design All** to save all the design data. **Close** the LAYOUT session.

Congratulations! You have completed the "Automatic Routing" lab exercise. Continue with Lesson 3: "Routing with the Dynamic Editor".

Lesson 3
Routing with the Dynamic Editor

This discusses how to use the Dynamic Editor to perform interactive routing on your design. The Dynamic Editor is one of the interactive routing tools in LAYOUT.



Objectives

In this part of the routing module you learn the Dynamic Editor. You use the Dynamic Editor to interactively route sketched traces. You also look at its ability to quickly shift traces to route in tight areas and to jump traces over pins as necessary to complete a route.

The Dynamic Editor is a tool for interactive routing in LAYOUT. It provides an alternative method of interactive routing for certain designs; it does not replace the other interactive routing capabilities of LAYOUT.

After completing this module, you are able to:

- Open the Dynamic Editor.
- Name the conditions that prohibit using the Dynamic Editor.
- Describe the effects of changing pad entry rules.
- Route traces using the Dynamic Editor.
- Return to LAYOUT.

Purpose of the Dynamic Editor

The Dynamic Editor is a specialized interactive router used for completing unroutes left by the autorouter. You sketch the path for the new trace to follow. The Dynamic Editor then attempts to route the trace, rerouting other traces in the path. This rerouting ability is the strength of the Dynamic Editor. It moves impeding traces to create a channel for the new trace. This task is quite tedious with a traditional interactive editor.

The following sections discuss how to enter the Dynamic Editor, as well as the unique terminology and characteristics of the Dynamic Editor.

Opening the Dynamic Editor

There are restrictions on the types of designs that can be interactively routed with the Dynamic Editor. You cannot use the Dynamic Editor on multiwire designs, orthogonal-only routing, a customized routing grid, or antennas. For designs with these restrictions, you must use the interactive routing features of LAYOUT. Despite these restrictions, the Dynamic Editor is ideally suited for many through-hole and SMD designs.

To open the Dynamic Editor, choose **Open Dynamic Editor** from the edit window popup menu.

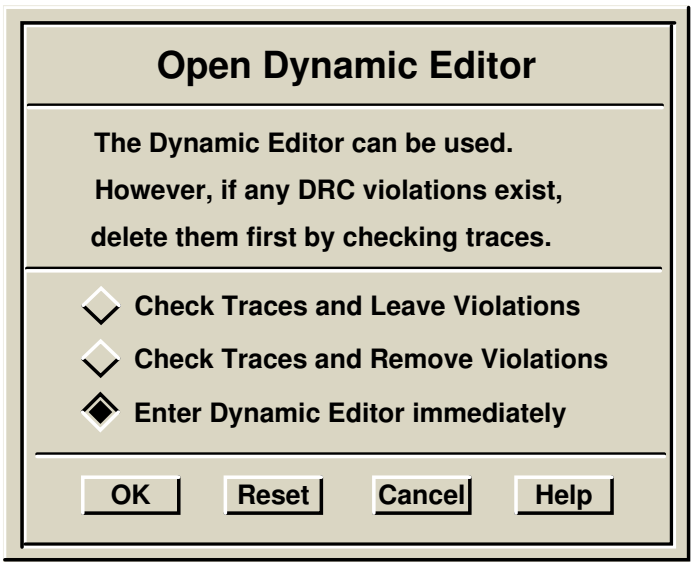


Figure 3-1. Open Dynamic Editor Dialog Box

The Dynamic Editor can be opened on a design whether or not design rule violations are present. If violations are present, the editor opens in read-only mode. The options for possible design rules violations are:

Check Traces and Leave Violations—Reports design rule violations. If violations are found, an error message is displayed and the editor does not open.

Check Traces and Remove Violations—Removes routing violations before opening the Dynamic Editor. The editor opens and all editing functions are available.

Enter Dynamic Editor Immediately—Opens the design without checking the status of routing. If violations are present, the editor opens in read-only mode.

Current Design Characteristics

There are a number of design characteristics that are currently incompatible with the Dynamic Editor and whose presence in your design prevents you from invoking the editor. Some of these, such as Multiwire board technology, are absolute. Others, such as using diagonals and t-junctions, are under your control and can be set to an acceptable state to allow use of the editor.

The following table groups incompatible design characteristics into three categories, and provides a brief description of the characteristics in each category.

Table 3-1. Incompatible Design Characteristics

Board Technology	Multiwire
Design Rules	Diagonals Not Allowed T-junctions Not Allowed
Miscellaneous	Stubs (Antennae) Exist on Board Trace endcode style other than round Design Rule Violations

In addition to the design characteristics listed in the table, there are design elements that can affect performance in the Dynamic Editor. They are:

- Large, complex fill areas
- A large number of fill areas
- Padstacks using multiple, non-concentric shapes

None of these elements disable invocation of the Dynamic Editor. However, to obtain optimum performance, do not invoke this editor on a design in which any of these elements are present.

DRC violations are not allowed when invoking the editor. It is recommended that you first remove DRC violations from your design.

Concepts and Terminology

Traces versus Segments and Vertices—A Dynamic Editor trace consists of all segments and vertices included between any two consecutive routing elements. Each is individually selectable and can be manipulated in several ways. The segments and vertices of LAYOUT do not exist to the Dynamic Editor.

Sketching—The interaction between you and the Dynamic Editor that guides the router around immovable objects, such as pins, vias, and tacks.

Tacks—A routing element that the Dynamic Editor uses to control traces. Tacks are found where a guide joins a trace, a guide joins another guide, a t-junction forms, a segment of one width joins a segment of another width, a protected segment joins an unprotected segment, and a trace reverses direction.

Live Trace—To the Dynamic Editor, all unprotected traces are "live"; or any trace might be affected by creating or altering any other trace. This means that in the process of choosing a path, the editor might reroute nearby traces to achieve the best route.

Movable versus Immovable Objects—To the editor, the trace is movable, while its pins, vias, and tacks, as well as those nearby, are immovable. You can manually move pins, vias, and tacks while using the Dynamic Editor.

Layer Directionality—In LAYOUT, the interactive editor allows routing in any direction on any layer. In the Dynamic Editor, each routing layer has an associated routing direction.

The Topological Box

The Dynamic Editor observes a constraining area called the topological box. This occurs whenever you are moving a via or tack. The topological box is a rectangle with sides that are defined by the routing elements (pins, vias, tacks) that define the trace or traces connected to the via or tack being moved.

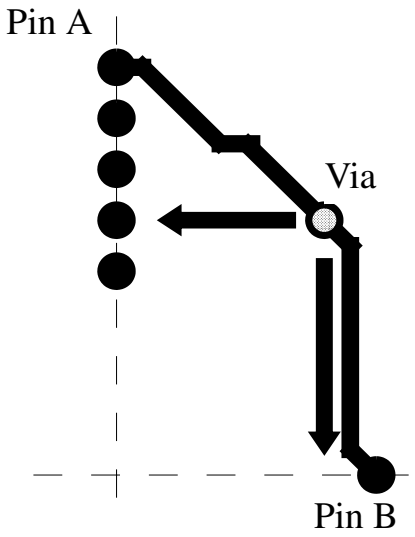


Figure 3-2. Topological Box

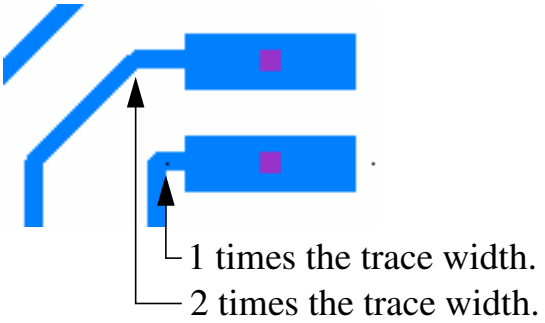
The arrows in the diagram show the limits of horizontal and vertical movement set by pins A and B. Attempting to move the via horizontally beyond the dashed line violates the directionality constraint. Also, the trace connecting to pin B is on a vertical layer, so vertical movement of the via below the dashed line violates the vertical directional constraint.

Specifying Pad Entry

You can specify how pad entry is accomplished. The options include:

- Pad entry/exit distance.
- Entry/exit bend snap to trace grid.

Distance set at:



The nature of the trace created when the editor enters or exits a pad can be further controlled by specifying whether the editor conforms to pad entry rules. The Dynamic Editor provides the following options for pad entry setup:

- Enter and exit pads only when pad entry rules can be satisfied.

  Depending on the nature of the design this can result in the editor failing to satisfy entry or exit rules for some pads, but guarantees that all connections to pads satisfy the rules.

- Try to satisfy pad entry rules, but always complete the trace.

  This guarantees that the editor satisfies pad entry rules wherever possible, but makes the connection even if it results in an invalid entry into a pad.

- Disregard pad entry rules.

  This mode allows the editor to choose the easiest pad entry direction, but it can result in a larger number of unsatisfactory connections than can occur with another option.

Toggling Pad Entry

For any pad shape and combination of trace width, clearance, and trace grid presence, one or more configurations are possible. The editor provides a facility for cycling through the possible entry/exit configurations for a given pad.

1. To toggle pad entry/exit choose the **Dynamic Editor > Toggle Pad Entry** menu item from the edit window popup menu.

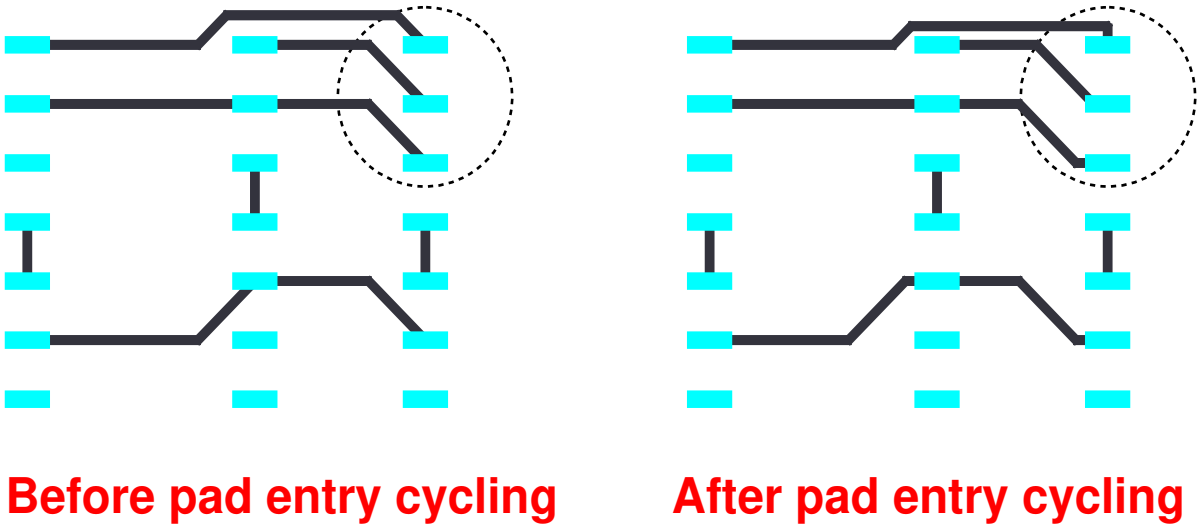


Figure 3-3. Pad Entry Options

This opens the Toggle Pad Entry prompt bar and changes the cursor to a large crosshair. By default, the function is set to repeat.

2. Point to the trace on the pad for which you want to cycle through the entry/exit options and click the Select mouse button.

   The trace is rerouted to enter/exit the pad to conform to one of the available configurations. Because the entire trace is rerouted, the opposite end, if it enters a pad, via, or tack, can also be reconfigured to conform to pad entry/exit rules.

3. Repeat step 2 to cycle to another entry/exit configuration.

   This process can be repeated to cycle through all valid pad entry configurations that are possible for the intended pad and trace.

4. To cycle through pad entry options on another pad, move the cursor to the desired pad and repeat from step 2.

5. When you are finished, press **Close** in the prompt bar.

Figure 3-3 shows an example of pad entry configurations. On the left, the three circled pads show traces routed without a trace grid and without regard to pad entry rules. These pad entry configurations are invalid. On the right, the three circled pads show the resulting configuration after one, two, and three cycles, respectively. After the third cycle, the pad entry configuration returned to the first cycle.

Routing with the Dynamic Editor

To route a trace with the Dynamic Editor, use the mouse to quickly draw a sketch of the basic path for the trace to follow through the immovable objects (pins, vias, and keepouts) that lie between the trace's target and destination. The Dynamic Editor then creates the trace based on the current design rules.

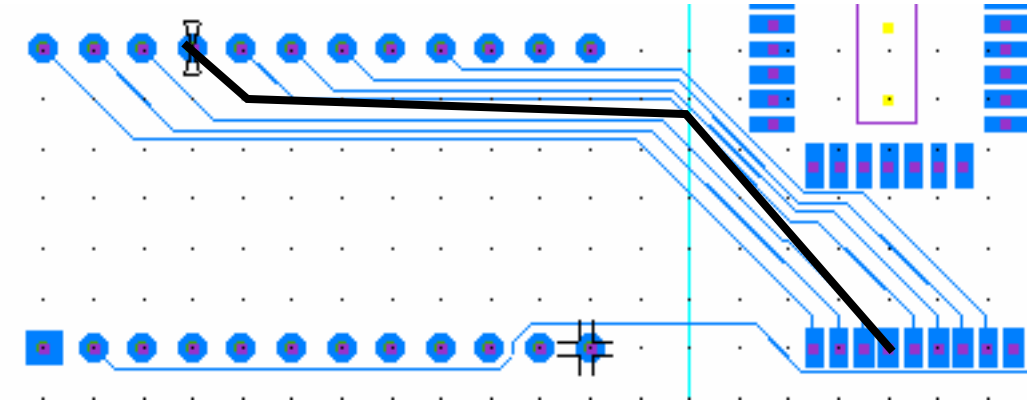


Figure 3-4. Sketching a Trace

If an existing trace blocks the path for the new trace, the existing trace is moved aside during a reroute to create room for the new trace, unless doing so causes a design rule violation or requires the existing trace to be rerouted to the other side of an immovable object. You can also reroute an existing trace to create a channel for a new trace; the Dynamic Editor automatically reroutes other affected traces if necessary.

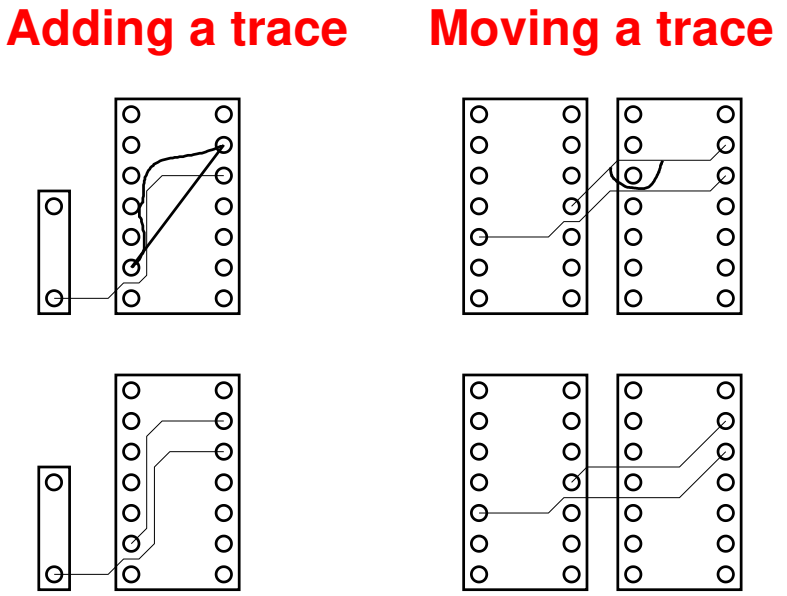


Figure 3-5. Adding and Moving Traces

Jump mode gives the Dynamic Editor even more flexibility. Setting jump mode to on allows the Dynamic Editor to reroute traces over immovable objects, such as pins and vias.

Creating Tacks

There might be situations that prevent you from making the trace image follow the cursor all the way to the connecting pin. Try making the trace image go as far as you can, and click the Select mouse button to create a tack.

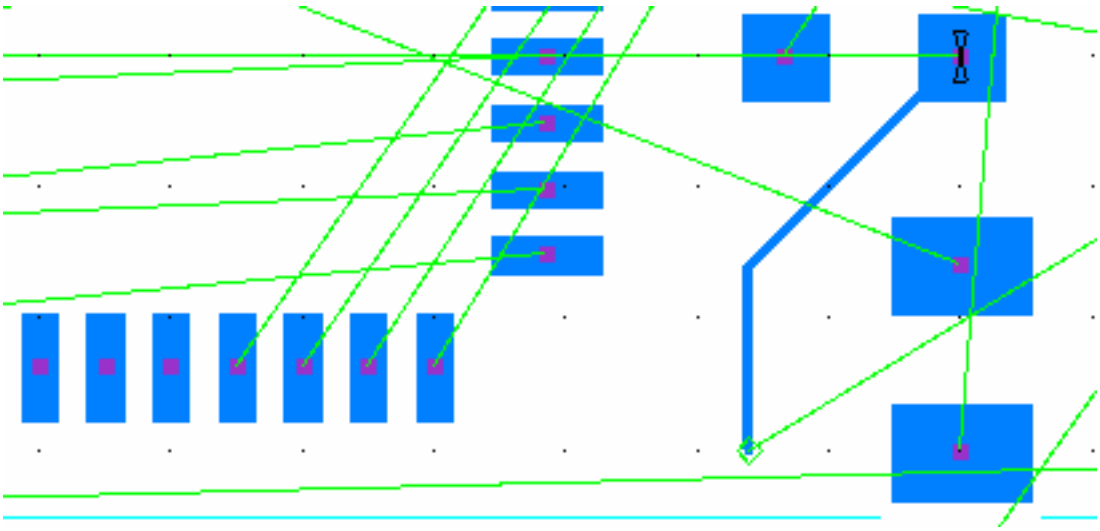


Figure 3-6. Continue Routing from the Tack

Sometimes several tacks are needed between two connecting pins. Other times, you must drop a via, and continue routing on another layer. It is a good idea to keep the traces on any one layer oriented in one direction.

Returning to LAYOUT

After you complete design work with the Dynamic Editor, choose the **File > Close Dynamic Editor** menu item. The Dynamic Editor presents a dialog box to handle the edits you have made while the Dynamic Editor was open.

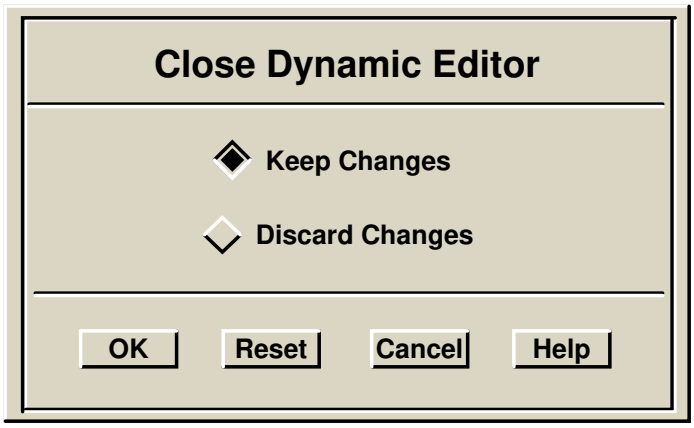


Figure 3-7. Close Dynamic Editor Dialog Box

Keep Changes—Choose this button if you want to retain all of the routing accomplished with the Dynamic Editor since you last opened it.

Discard Changes—Choose this button if you want to discard all of the routing you did with the Dynamic Editor since you last opened it.

Lab Exercise

This lab exercise gives you a chance to use the Dynamic Editor in a variety of interactive routing situations.

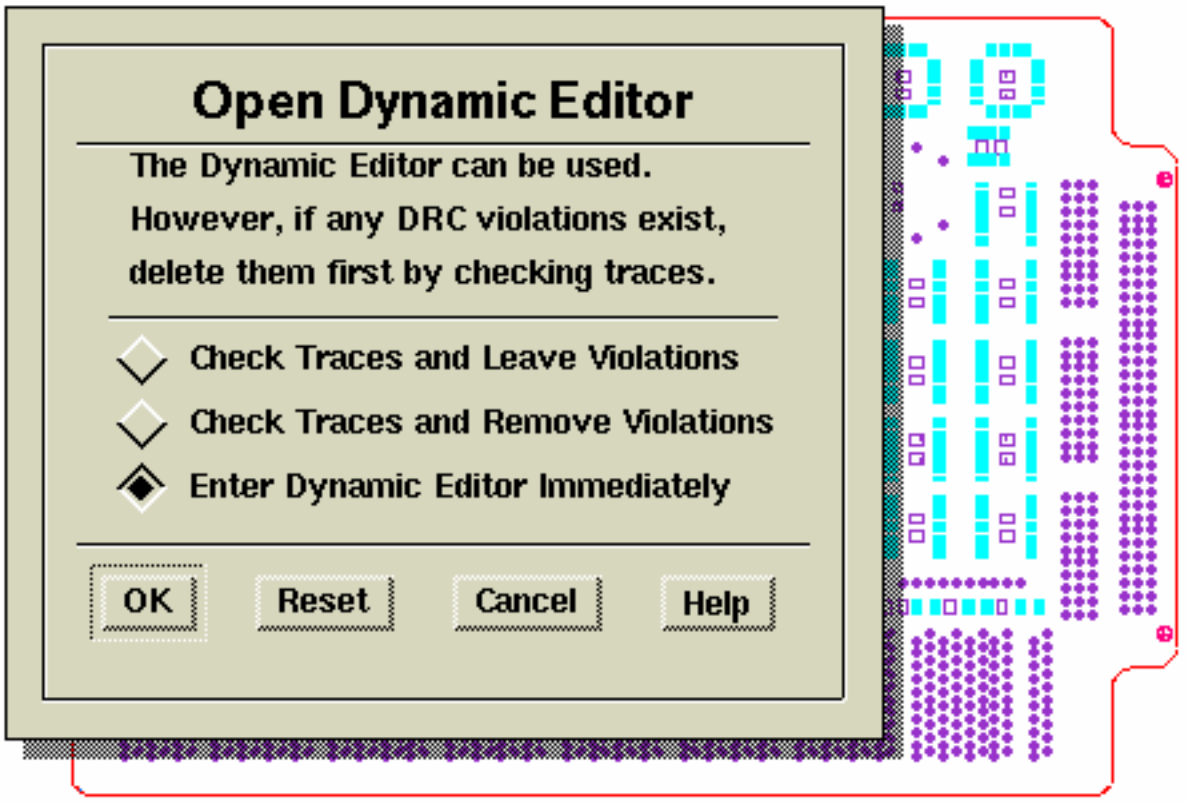
Upon completion of this lab exercise you are able to:

- Set up your lab design for the Dynamic Editor routing.
- Open the Dynamic Editor.
- Route traces interactively.
- Return to LAYOUT and autoroute traces.
- Complete routing using the Dynamic Editor.

Turn to Module 6—Lab 3: "Routing with the Dynamic Editor".

Lab 3
Routing with the Dynamic Editor

This lab exercise familiarizes you with interactive routing using the Dynamic Editor.



Introduction

This lab exercise gives you a chance to use the Dynamic Editor in a variety of interactive routing situations. You use the Dynamic Editor to pre-route traces and to complete traces after the board has been autorouted.

Upon completion of this lab exercise you are able to:

- Set up your lab design for the Dynamic Editor routing.
- Open the Dynamic Editor.
- Route traces interactively.
- Return to LAYOUT and autoroute traces.
- Complete routing using the Dynamic Editor.

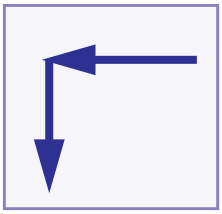
Procedure

In this lab exercise, you use the Dynamic Editor to interactively route traces on the board.

Preparation for Lab

You start by invoking LAYOUT and then delete all the traces. Remember, you saved the traces in the previous lab, so you can restore them, even after you delete them.

1. Invoke the Design Manager. Next, invoke LAYOUT on the **sig_az** design using the LAYOUT icon in the Tools window.
2. After LAYOUT invokes, close the report window.



3. Set the Select Filter to allow selection of Traces, Vertices, and Vias.
4. Choose the **[Top Menu] Auto Routing > Select > Select Traces** menu item. Select the traces connecting to the connector in the analog section of the board. Delete the selected traces.

   It is helpful in this lab exercise to view the traces in their actual width.

   The traces are replaced with guides.
5. Choose menu item **View > Change View Style**. In the dialog box, select **Filled** as the Polygon/Trace Style and **Stroked** as the Text Style. Then select **OK**.

Opening the Dynamic Editor

Before opening the Dynamic Editor it is good practice to check the compatibility of your design. Sometimes user-definable design rules prevent opening the Dynamic Editor.

1. Choose the **Report > Dynamic Editor Status** menu item.

   The message window states that the current design rules are not compatible with the Dynamic Editor because the Dynamic Editor requires design rules that allow diagonal traces.

2. Set up the routing rules to allow diagonal traces, then get another report on the Dynamic Editor status.

   The note in the message window now states that the Dynamic Editor can be used. Next you open the Dynamic Editor.

If the status report advises you of any violations, you must fix the violations before entering the Dynamic Editor. If violations exist, the Dynamic Editor opens in read-only mode.

3. Choose the **[Top Menu] Open Dynamic Editor** menu item. In the Open Dynamic Editor dialog box, choose **Enter Dynamic Editor Immediately**, and then **OK** the dialog box.

   The Dynamic Editor dialog box offered three choices for handling entry into the Dynamic Editor. The appropriate choice depends on how you want to handle existing traces and violations in routing. You chose to Enter the Dynamic Editor Immediately.

The contents of the session window redraw and all the menus are replaced with Dynamic Editor menus. LAYOUT is now in the Dynamic Editor mode, and remains so until you exit the Dynamic Editor.

Interactive Routing Setup

The view in the Edit window is the same as in LAYOUT. Environment settings, such as viewable layers, also remain unchanged. To demonstrate the Dynamic Editor features, set up grids, pad entry/exit, and toggle layers.

1. Choose the **Setup Routing > Trace Grid...** menu item. In the dialog box, set the x and y values to **0.02**, and then **OK** the dialog box.

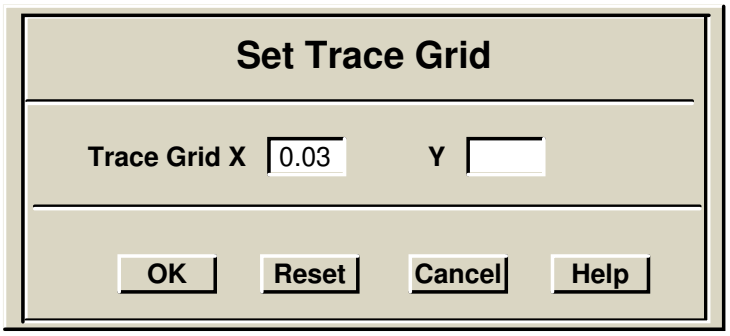


Figure 3-8. Setting the Trace Grid

You can also route with the Dynamic Editor in a gridless mode by setting a value of 0. But, because some routing on the board is complete, use the Dynamic Editor in a gridded mode.

You can control the behavior of the Dynamic Editor when it enters or exits a pad. Controlling the behavior of the Dynamic Editor means specifying the distance from a pad to the first bend and whether the bend point snaps to a grid.

2. Choose the **Setup Routing > Set Pad Entry Rules...** menu item. In the dialog box, choose Set Pad Entry Rules to: **On**, then **OK** the dialog box.

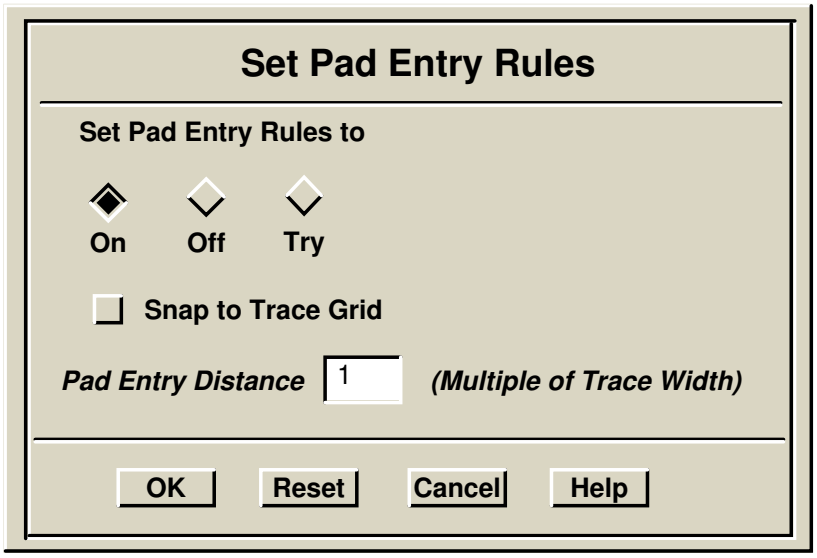


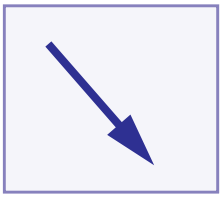
Figure 3-9. Setting Pad Entry Rules

This choice causes the Dynamic Editor to leave one trace width between a pad and a bend on entering or exiting the pad.

3. Choose the **Setup Routing > Edit Layer...** menu item. In the dialog box, click the Select mouse button on the layer names until only Trace_Layer_1 and Trace_Layer_2 are highlighted with a *T* next to them. In the left half of the dialog box, click next to Trace_Layer_1 so *Edit Layer* displays next to that layer. **OK** the dialog box.

   Now when you route, the initial Edit layer is Trace_Layer_1, and you can only toggle the routing layers between Trace_Layer_1 and Trace_Layer_2. So, when you create a via, the route can only go to Trace_Layer_2. When you click again on a via to change the routing layer, the via is removed because the routing layer is returned to the top layer of the board.

Routing Traces Interactively



You are going to route all of the connections to the connector on the left side of the board.

1. Use the View Area stroke to view the area around the analog section, including the section of the connector to the left of the analog area.

   The following instructions guide you through creating your first trace.

2. Choose **Route Trace** from the Dynamic Editor palette menu.

   The Route Trace prompt bar displays, and prompts you to enter a point. The cursor in the edit window changes to the location crosshair cursor.

3. Place the cursor on a guide line near where it connects to a pin, and click the Select mouse button. Move the cursor and notice how the ghost image of the trace moves with the cursor.

   The guide is selected, and the image of the trace moves with the cursor. If you move the cursor along an unavailable path, the trace does not follow the cursor. If you move the cursor in a direct line across the board, the trace image automatically follows a path that goes around other traces and component pins.

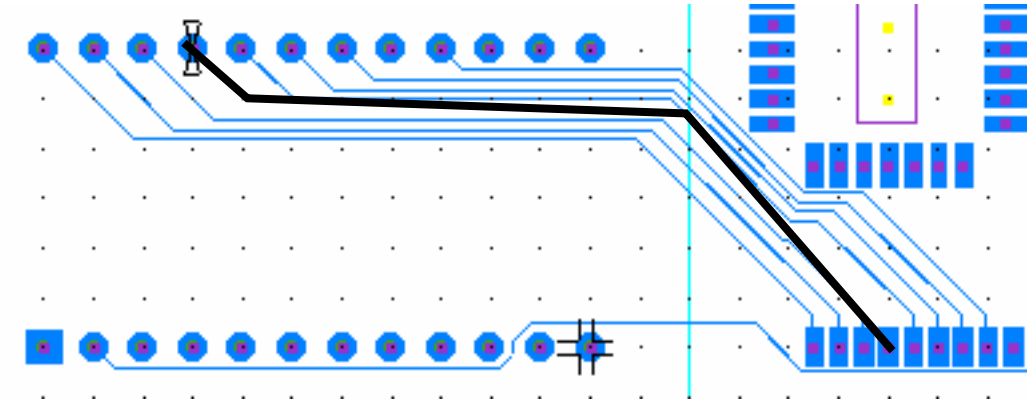


Figure 3-10. Sketching a Trace

4. Move the cursor in the general direction of the guide toward the pin to which it connects. Place the cursor on the pin at the end of the guide line, and click the Select mouse button to complete the trace.

Creating Tacks

1. If you cannot make the trace image follow the cursor to the connecting pin, and try to make the trace image go as far as you can, and click the Select mouse button to create a tack.
2. Continue routing the trace from the tack.

In the Dynamic Editor, a vertex is also called a tack. *A tack holds a point of the trace in a fixed position while you route the rest of the trace.*

Sometimes several tacks are needed between two connecting pins. Other times, you must drop a via, and continue routing on another layer. It is a good idea to keep the traces on one layer oriented in one direction. The Dynamic Editor routes best in the routing direction specified in the layer rules for the Edit layer.

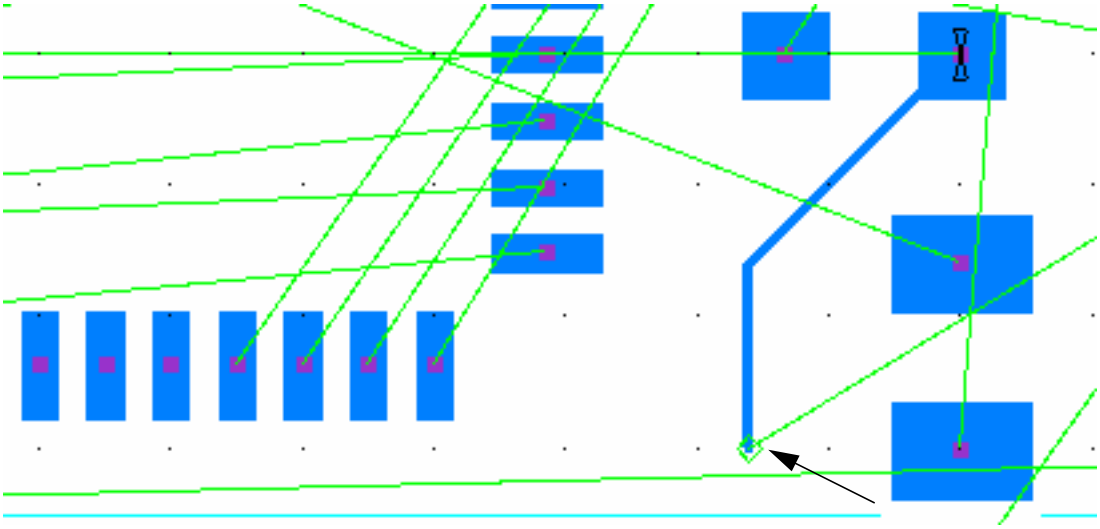


Figure 3-11. Guide Continues from a Tack

You set up the layer rules in the lab on Design Rules and Interactive Routing. In this design the Trace_Layer_1 layer has a horizontal routing direction, and the Trace_Layer_2 layer has a vertical routing direction. You can verify this by checking the Edit layer name displayed in the status window. If Trace_Layer_1 is shown, the letter *H* displays to its right. When you create a via, thereby changing the edit layer to Trace_Layer_2, the layer displays the letter *V*.

Changing Layers

The technique for adding vias has not changed with the Dynamic Editor.

1. As you route a trace, click the Select mouse button twice. The editor places a via and sets your edit layer to the next toggle position.

   You can also add a via by using the **[Dynamic Editor] Add > Add Via to Trace** menu item.

   If you change your mind about routing a trace after you have begun to drag it, you can unselect it by pressing the Unselect All function key. After you unselect a trace, you can select another guide for routing.

2. Complete routing a trace.

   The Route Trace prompt bar prompts you to create another trace.

Re-routing a Trace

In the next step, you reroute a portion of an existing trace. Previously drawn traces can be in an inconvenient location, and you must reroute them so other traces can be created. The next step demonstrates rerouting traces.

1. When the Route Trace prompt bar asks you for a new point, place the cursor on an existing trace at a point where you would like to start a change in its path, and click the Select mouse button. Move the cursor so you see the sketch line showing the rough new path for the trace. Move the cursor so the sketch line follows a different path than the existing trace, and then reconnects to the trace closer to one of connecting pins. Click the Select mouse button.

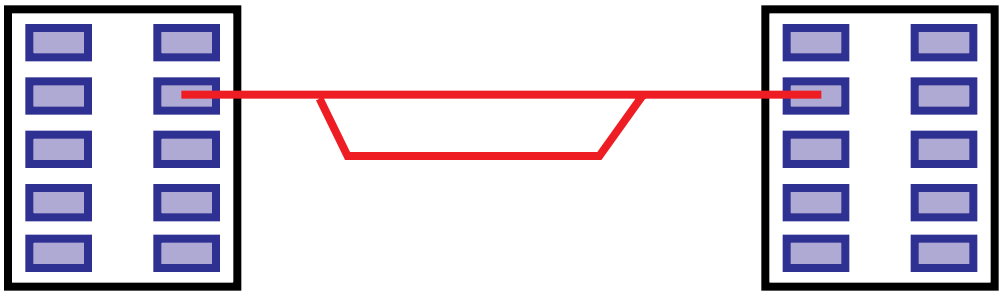


Figure 3-12. Defining a New Route for a Trace

The trace reroutes following the route you sketched as closely as design rules allow. If there are other traces close to the new path you choose, the editor shoves neighboring traces aside to make

room for the rerouted trace. The editor might also jump pins and vias, if necessary, to accommodate the new path.

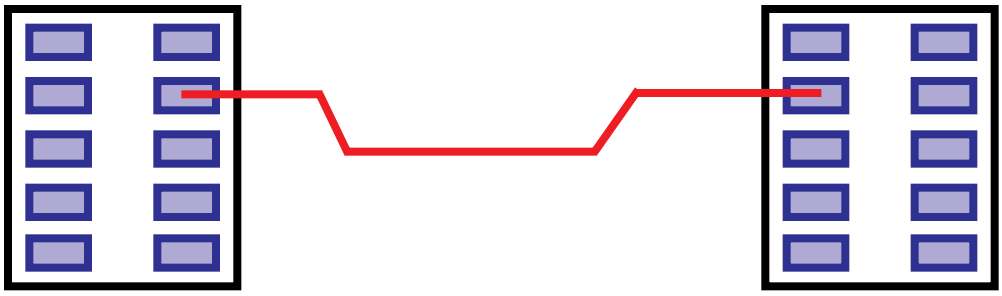


Figure 3-13. The Rerouted Trace

2. Continue creating traces. Make sure you place some vias in the traces.

You need at least one via for the next step, where you learn how to move a via or tack.

Moving a Via

1. Choose the **[Dynamic Editor] Move** menu item.
2. Place the cursor on a via and click the Select mouse button. Move the cursor to see the ghost image of the via. Move the ghost image to another convenient location and click the Select mouse button again.

   The via moves and the prompt bar repeats so you can move another via.
3. **Cancel** the Select Via prompt bar. You can also cancel any other prompt bars that might be displayed.

Moving a Tack

Next, try moving a tack. Remember, a tack is a fixed point in a trace you create when you click the Select mouse button at any point along a guide path.

1. Choose the **[Dynamic Editor] Move** menu item.

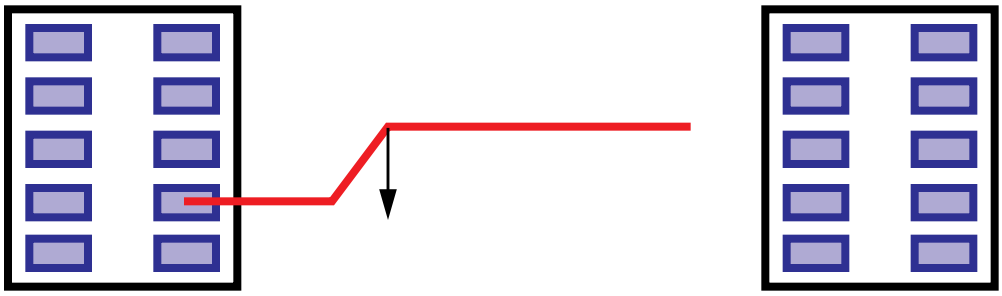


Figure 3-14. Drag the Tack

2. Place the cursor on a tack you created in a trace and click the Select mouse button. Drag the ghost image of the tack to a new location and click the Select mouse button again.

Figure 3-15. Result of Moving the Tack

The move fails if the move results in a violation or if nearby traces would have to be shoved. You can only move tacks you create. You cannot use this technique to move other bends in traces. To move system-made bends in traces, you could reroute the portion of the trace that includes the system-made bend.

3. **Cancel** the prompt bar.

Deleting a Trace

Next, you delete a trace.

1. Choose the **[Dynamic Editor] Delete** menu item.

2. Place the cursor on a trace you want to delete and click the Select mouse button. In the dialog box, choose **OK** to delete the selected trace.

3. **Cancel** any prompt bars that might be displayed.

Selecting a Pad Entry

For a given pad shape and combination of trace width, clearance, and trace grid settings, one or more legal pad entry and exit configurations might exist. You can cause the editor to cycle through the possible combinations. You try this in the next step.

1. View a small area around the end of a trace where it connects to a pin.

2. From the Dynamic Editor palette, choose the **Toggle Pad Entry** icon.

3. Place the cursor on the pin to which the trace connects and click the Select mouse button.

   The trace reroutes to one of the legal configurations. The prompt bar repeats by default.

4. Click the Select mouse button again.

   The trace reroutes to another legal pad entry/exit configuration. Because the prompt bar remains on the screen, you can move the cursor to another pad and cycle through legal pad entry options.

5. **Cancel** the prompt bar.

Complete routing the traces from the connector.

Continue routing the traces from the connector in the analog section of the board. It is not essential that you finish all the traces. In the next section, you use the autorouter to route most of the board. At the end of this lab, you use the Dynamic Editor to complete any traces the autorouter did not complete. Completing traces that the autorouter cannot complete is one of the best uses of the Dynamic Editor.

Returning to LAYOUT

1. When you complete routing the traces running from the connector, choose the **File > Close Dynamic Editor...** menu item.

2. Choose **Keep Changes** and **OK** the dialog box.

   You return to the normal mode of the LAYOUT application. Next you use the autorouter to complete the routing.

3. From the Auto Route palette, choose the **Auto Route** icon. In the dialog box, enter the following and **OK** the dialog box.

   Pattern

   Number of Passes: **2**

   Starting Cost Schedule Pass: **[Leave this box at default]**

   Automatic

   Number of Passes**: 3**

   Starting Cost Schedule Pass: **[Leave this box blank]**

   Manufacturing

   Number of Passes**: 2**

   Attempt to Route Guides

   Automatic [Leave the remainder of this section blank]

   Remove Intersections When Done

   Freeze Display

   Save Traces: **Never**

   Route Entire Board

4. When the routing completes close the report window.

Completing Routing Using the Dynamic Editor

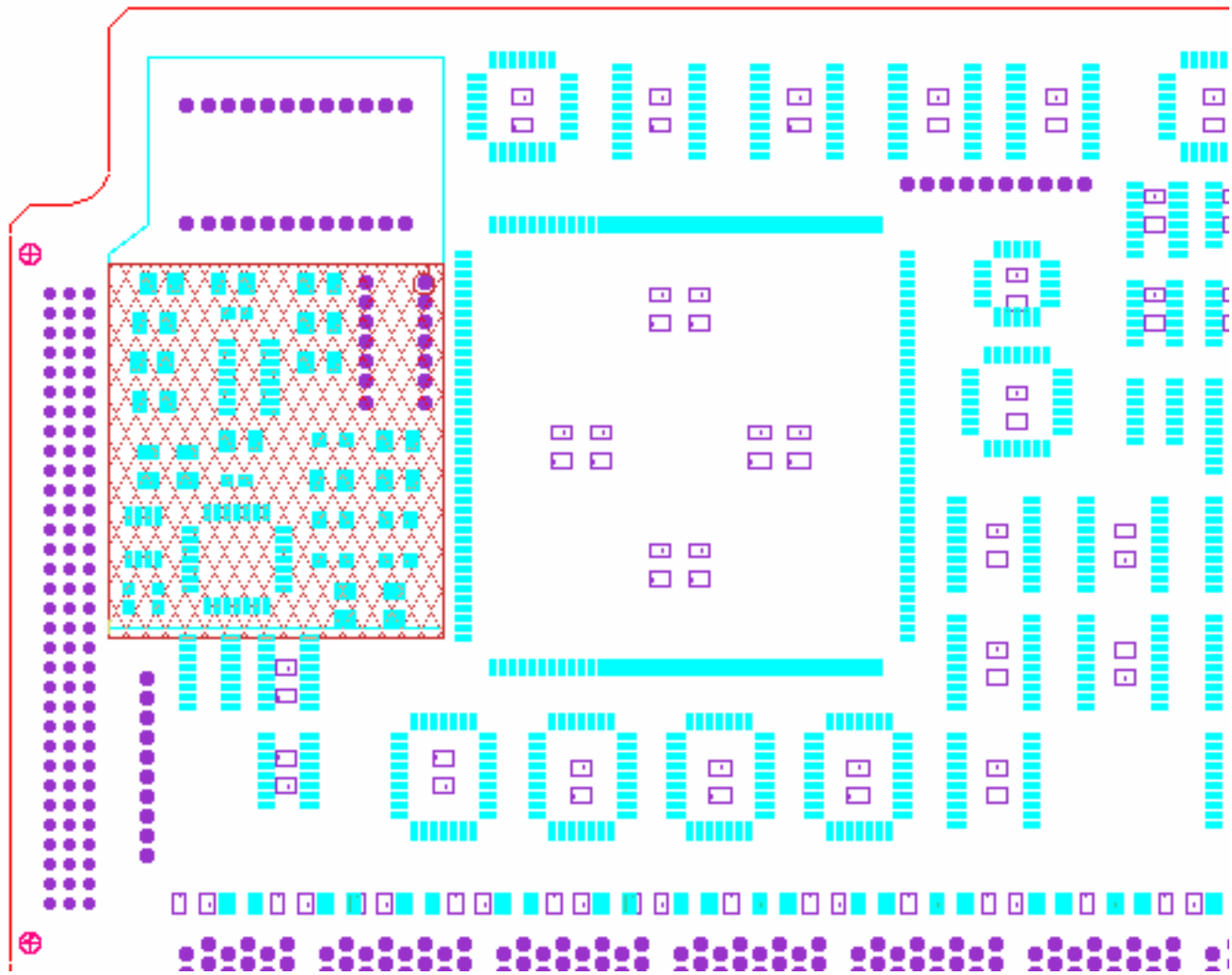
To finish this procedure, you return to the Dynamic Editor. Use the routing functions in the Dynamic Editor to complete your board.

1. Open the Dynamic Editor and complete all routes.
2. When you finish routing, close the Dynamic Editor, and keep all changes.
3. Save all the design data and close the LAYOUT session.

Congratulations! You have completed the "Routing with the Dynamic Editor" lab exercise. Continue with Lesson 4: "Creating Area Fills."

Lesson 4
Creating Area Fills

In this lesson for routing traces on a circuit board, you learn about adding area fills to your design.



Objectives

In this, the last part of the routing module, you examine another type of circuit board conductor area, the area fill. Area fills can be added to any signal or power layer. The area fills might be associated with nets in your design and can include cutout areas.

After completing this module, you should be able to do the following:

- Describe the characteristics of an area fill.
- Describe how net connectivity is handled for area fills.
- Name the steps necessary to create an area fill.

Area Fills

LAYOUT (and FabLink) provide features for creating area fills on both signal and power layers. Fills created on power layers are called power fills.

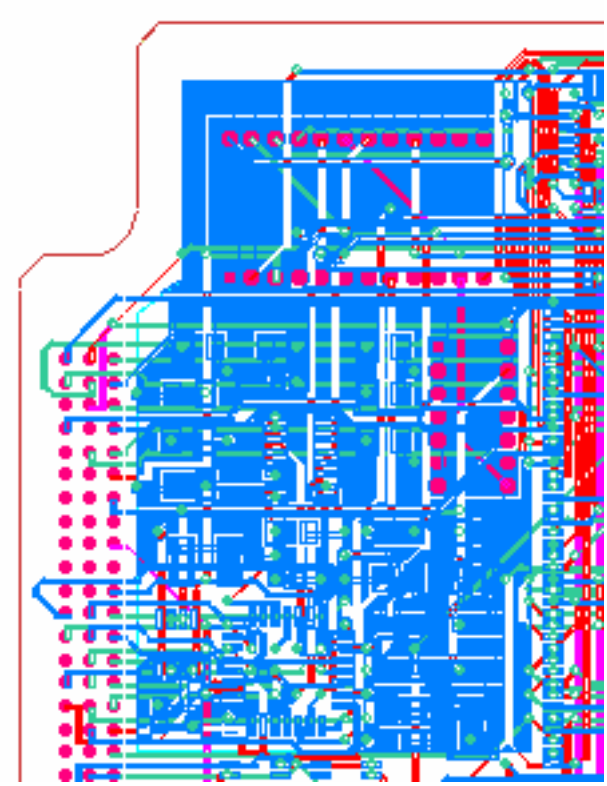


Figure 4-1. Area Fills May Associate with a Specific Net

You can specify options that create the fills as either solid areas or any of several patterns. You can also create a cutout within a fill to remove copper.

When you create a fill you can associate it with a particular net, or with no net association at all. Fills associated with a net provide connectivity within the net.

Area Fill Connectivity

Because an area fill has connectivity with respect to the net with which it is associated, it can be used to connect any two elements within the net. Another way of saying this is that an area fill can replace a guide. Conversely, an area fill that provides connectivity within a net is replaced with a guide when the area fill is deleted or cut away from an element within the net.

An area fill connection to a pin, via, or trace is recognized if the pin ID, via center point, or trace center line extends into the area fill to a distance equal to one-half the area fill aperture size. So, an area fill can connect two pins in a net, and replace the guide between them, if it encompasses the pins and their pads completely. However, if the pin ID of one or more pads lies on the edge of the area fill, the guide remains.

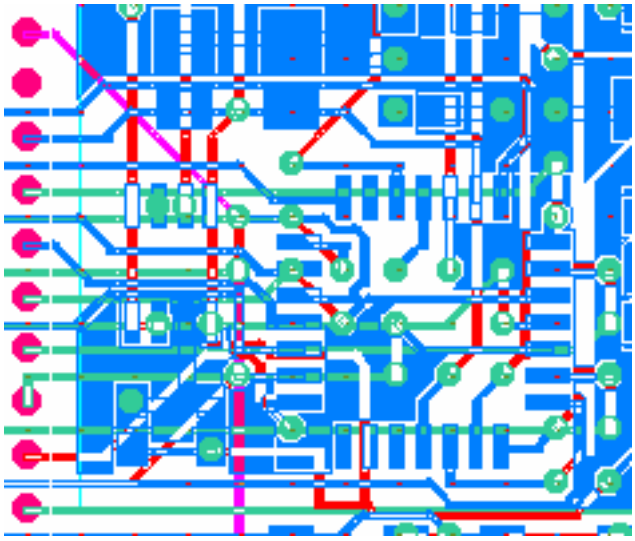


Figure 4-2. Automatic Checking On Maintains Connectivity

You must have Automatic Checking On when creating area fills to maintain correct connectivity. If a trace is part of the same net as the area fill, the fill connects to the trace. If the trace is a different net than the fill, a relief area is generated. Automatic Checking must be On for the relief areas to generate correctly.

Creating an Area Fill

In the Edit window, select the **Area Fill > Add Area Fill** menu item.

Figure 4-3. Add Area Fill Prompt Bar

Using the Add Area Fill prompt bar:

- **Net Name**—the name of the net to associate with the area fill. You can type a net name into the box, or point with the crosshair cursor to any pin, via, or trace in the desired net and click the Select mouse button.
- **Layer**—an entry box for the name of the signal layer on which you want to add the area fill. By default, the box displays the name of the current edit layer.
- **location button**—sets the Select mouse button to add points that define the area fill polygon in the Edit window.
- **Options**—a button you select to display the options dialog box that provides choices for various manufacturing parameters.

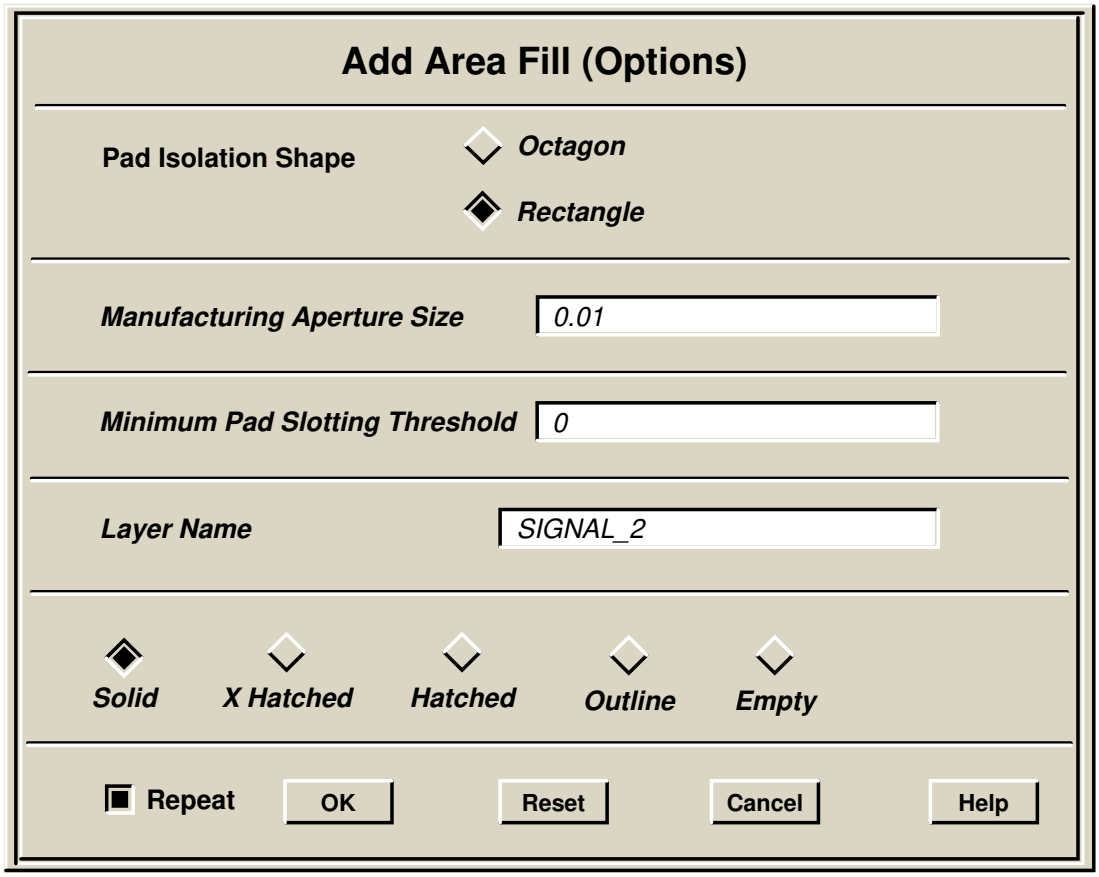


Figure 4-4. Add Area Fill Dialog Box

- **Pad Isolation Shape**—selects the shape of the cutout formed around pins and vias of nets not associated with the area fill.
- **Fill Pattern**—selects an artwork fill pattern for the fill area. *Outline* and *Empty* patterns act as trace keepouts for screened areas. *Outline* creates a permanent area. *Empty* creates a temporary area that is not manufactured.
- **Manufacturing Aperture Size**—specifies the aperture size used for area fill calculations and trace trimming.
- **Minimum Pad Slotting Threshold**—specifies the minimum distance, in user units, between any two pins for creating a slot between the pins, rather than isolating the pins individually. A value equal to the pin-to-pin distance ensures that slotting occurs.

Slotting Threshold

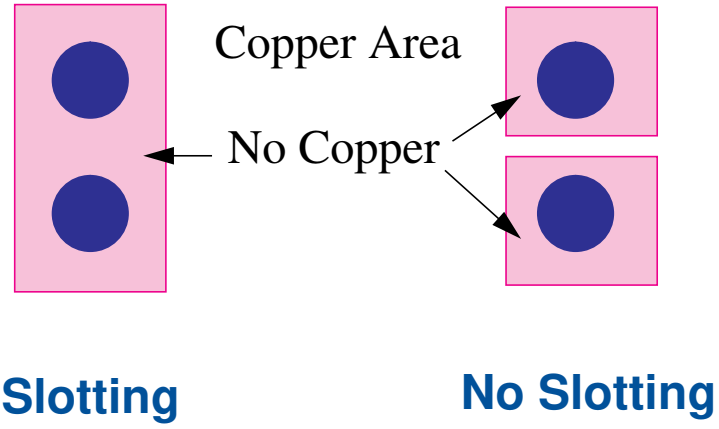


Figure 4-5. Set the Slotting Threshold

Slotting is the automatic generation of cutouts around pins. A line of pins are in a cutout slot. To generate slotting, enter a value equal to or greater than the pin pitch as the Minimum Pad Slotting Threshold. A value less than the pin pitch ensures no slotting occurs.

Summary

In this module you were introduced to four areas of printed circuit board routing; they are:

- the process of interactive routing and routing design rules.
- the techniques for routing with the Dynamic editor.
- examining the grids, costing, and operation of the automatic router.
- exploring area fills and their characteristics.

Lab Exercise

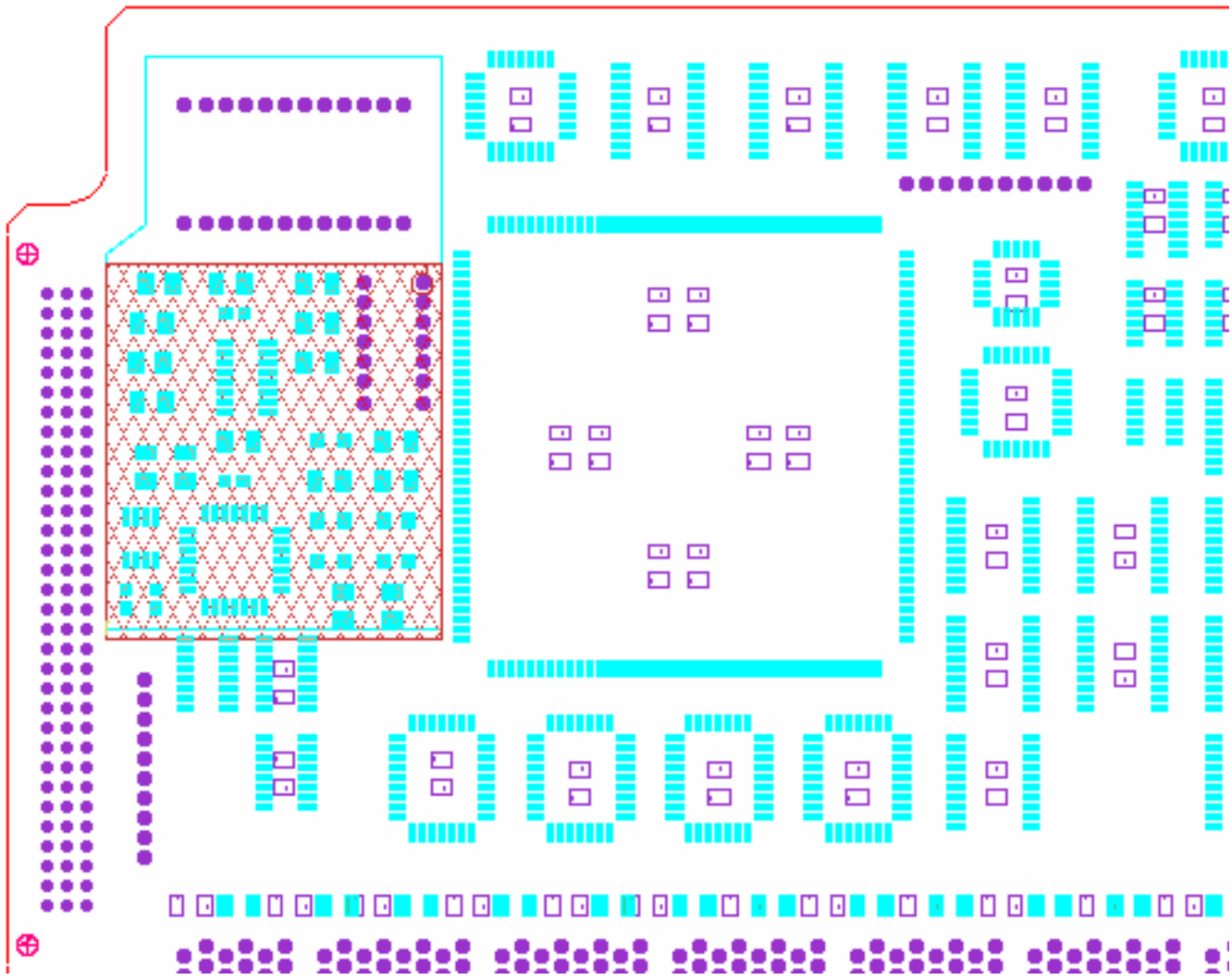
This lab exercise explores several aspects of creating area fills for ground plane or shielding on a circuit board. You create an area fill and a cutout in the area fill on the lab design board.

Upon completion of this lab exercise you are able to add area fills to your design.

Turn to Module 6—Lab 4: "Area Fills".

Lab 4
Creating Area Fills

This lab exercise familiarizes you with creating area fills and cutouts.



Introduction

This lab exercise explores several aspects of creating area fills for a ground plane or shielding on a circuit board. You create an area fill and a cutout in the area fill on the lab design.

Upon completing this lab exercise you can add area fills and cutouts to your design.

Procedure

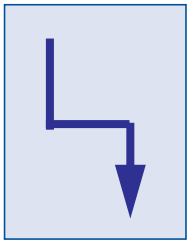
You use LAYOUT to create an area fill and a cutout in the area fill on the lab design.

Preparation for Lab

1. Invoke the Design Manager. Next, invoke LAYOUT on the **sig_az** design using the LAYOUT icon in the Tools window.

2. After LAYOUT invokes, close the report window.

Creating an Area Fill



In this procedure, you add a ground fill around the analog portion of the board.

1. Change the view area making the analog section of the board and some of the surrounding circuitry visible.

2. Choose **View > Highlight Nets > Highlight Nets**. Select the ***ground*** net from the list.

3. Choose the **[Area Fill] Add Area Fill:** menu item. In the Add Area Fill prompt bar, enter *Net Name:* ***ground***. Then choose **Options**. In the options dialog box, enter the following and **OK** the dialog box.

   Pad Isolation Shape: **Octagon**

   Manufacturing Aperture Size: ***0.01***

   Minimum Pad Slotting Threshold: ***0***

   Layer: ***SIGNAL_1***

   Fill Pattern: **Solid**

   In the next step, you specify a polygon for the area fill. The polygon you create will entirely enclose all the analog circuitry, and the polygon will be filled to create the ground shield.

   The fill_area prompt in the prompt bar has an icon that tells you more than one point is required as input.

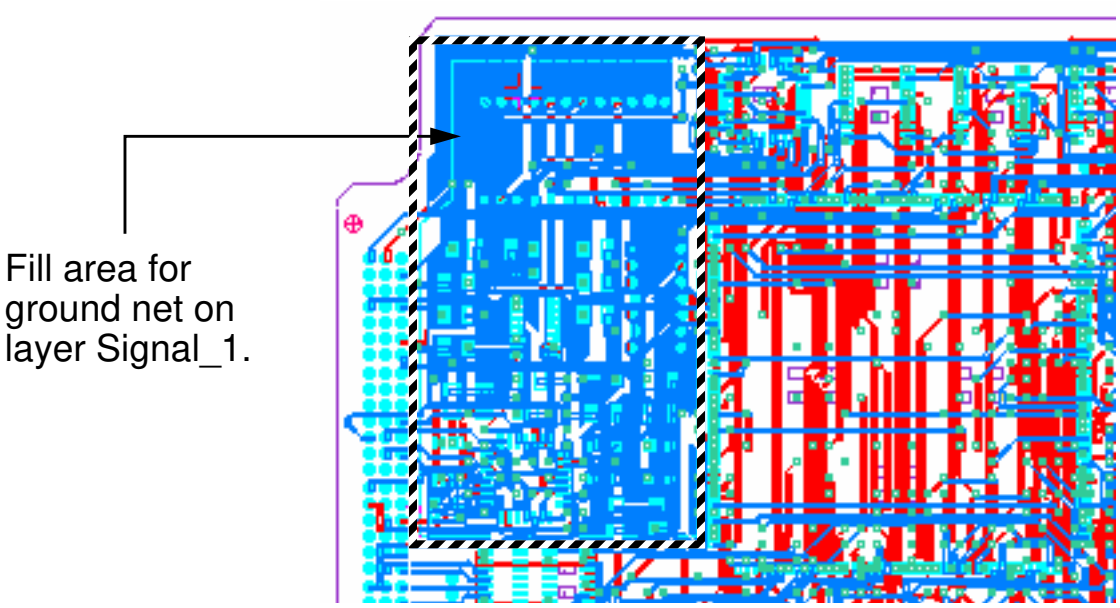


Figure 4-6. Adding a Fill Area for the Ground Net

4. In the prompt bar, either tab to the **fill_area** prompt, or click on it. Using the Select mouse button, click on the corners of a polygon to surround the area around the analog circuitry where a ground shield goes. Make sure the fill area you define is generously over-sized, so there is enough area within the fill area for you to create a cutout in the fill area later. After you click on the final vertex of the polygon, **OK** the prompt bar.

   A report window displays; it can contain several warnings about discarded fill areas, poor coverage for pins, and the fill area being fractured. The exact warnings you receive depend on the area you defined with the polygon, the arrangement of the components, and the proximity of other circuitry and traces to the fill area.

5. Close the report window and **Cancel** the Add Fill Area prompt bar.

   A polygon-like area represents the area fill. The fill pattern is only the graphic fill pattern associated with the display graphics. The actual pattern you choose (solid or hatched) does not display at this time.

Create a Cutout in the Area Fill

Place a cutout into the shield in case you later decide to place the board name or number on the board.

1. Choose the **[Area Fill] Cutout Area Fill:** menu item. In the prompt bar, enter the following information.

   Net Name: ***ground*** *Layer*: ***SIGNAL_1***

2. Tab to the location prompt or click on it with the Select mouse button.

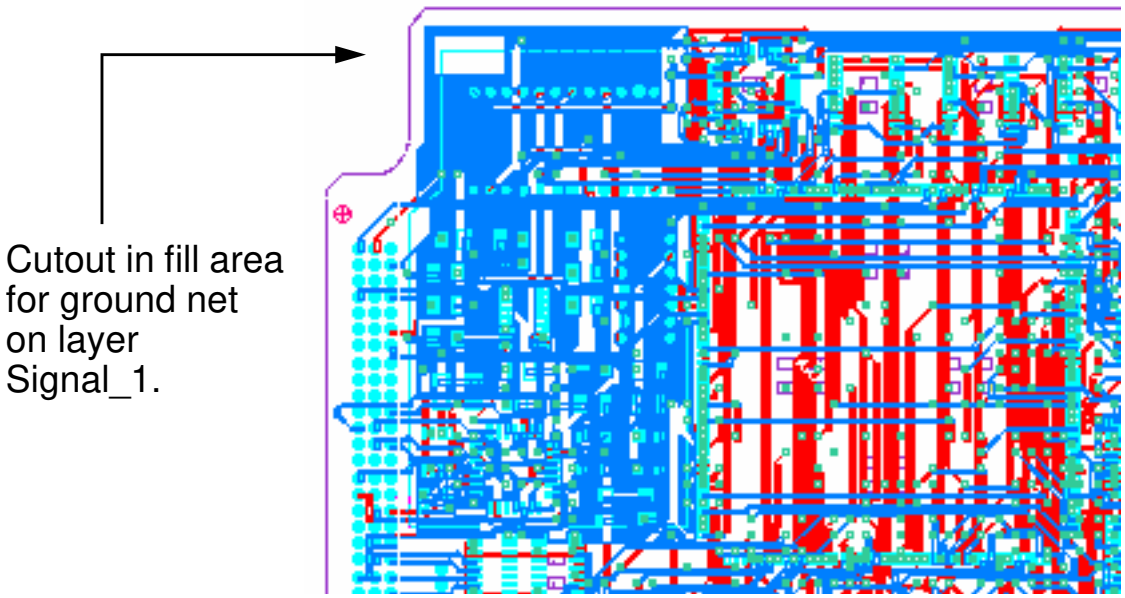


Figure 4-7. Adding a Cutout in a Fill Area

3. Using the Select mouse button, click on the vertices of a polygon that identifies a small area to cut out within the area fill. Use the cutout shown in Figure 4-7 as a guide. **OK** the prompt bar.

   The area you just defined is removed from the area fill, making it look as if the area has been cutout. When the board is fabricated, this area is etched back to bare board unless you put something in this area.

4. **Cancel** the Cut Area Fill prompt bar.

Experiment with Area Fills

If you want to cut out another area, and there is enough room in the area fill, you can create another area cutout. Trim the edge of an existing area fill by overlapping the edge of the area fill with a cutout area.

1. Create two more area fills. Make the second area fill overlap the existing area fill.

   The system merges the two fills when they overlap and have the same rules. If the fills use different rules, then they do not merge. If the fills belong to two separate nets, the fills stay separate with the trace clearance between them.

2. If there is enough room in the area fill, try creating another area cutout. Trim the edge of an existing area fill by overlapping the edge of the area fill with a cutout area.

3. When you finish, delete all but the first fill area and cutout you generated in the analog section of the board, as shown in Figure 4-7.

Back Annotate

If you add or change any properties or component locations while routing the design, you need to back annotate. It is good design practice to back annotate when you complete routing.

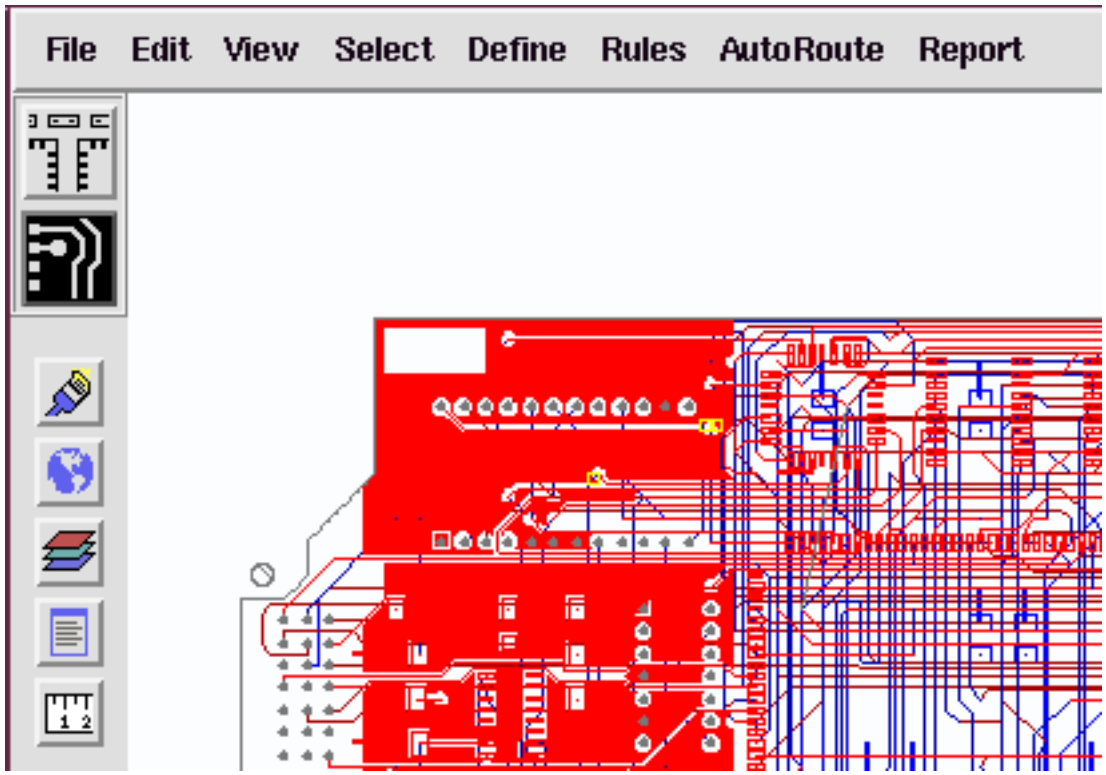
1. Choose menu item **File > Back Annotate**.

2. Save all the design data and **Close** the LAYOUT session.

Congratulations! You have completed the "Creating Area Fills" lab exercise. Continue with Lesson 5: "Shape-based Autorouting".

Lesson 5
Shape-based Autorouting

In this module, you use the SPECCTRA shape-based autorouter to enter basic routing commands and gain an understanding of shape-based routing concepts.



Objectives

This lesson explains the concept of shape-based autorouting and the organization of *do* files. It explores the following areas:

- Shape-based router process and environment
- Shape-based router concepts
- Purpose of do files and do file commands

The SPECCTRA Router in the MGC Environment

The easiest way to invoke the shape-based router is from within a LAYOUT session. Using this method automatically creates the files required to translate a Board Station design into a SPECCTRA design file, and after routing, creates the files used to update appropriate design objects in LAYOUT. Although you can invoke the shape-based router using the Design Manager or from the command line, this lesson focuses on invoking it from LAYOUT.

Figure 5-1 illustrates the translation steps automated by invoking the shape-based router from a LAYOUT session. The heavy arrows represent both the intermediate files and the translation required to create the SPECCTRA design input file and to interpret the *routes* output file. The SPECCTRA design file is shaded, because it is the final step of the automatic translation of your design.

When you invoke the shape-based router from LAYOUT, the system supports good data management by creating a directory in the *pcb* container in which to store all the intermediate, input, and output files generated for and while using the shape-based router. Creating a place to store all relevant files encourages you to keep the SPECCTRA version of your design in sync with the Board Station version, by facilitating your ability to monitor creation dates at the operating system level. You manage these files from LAYOUT. However, once all files are in place, it is your responsibility to create new data files when you change the Board Station design and use the shape-based router again.

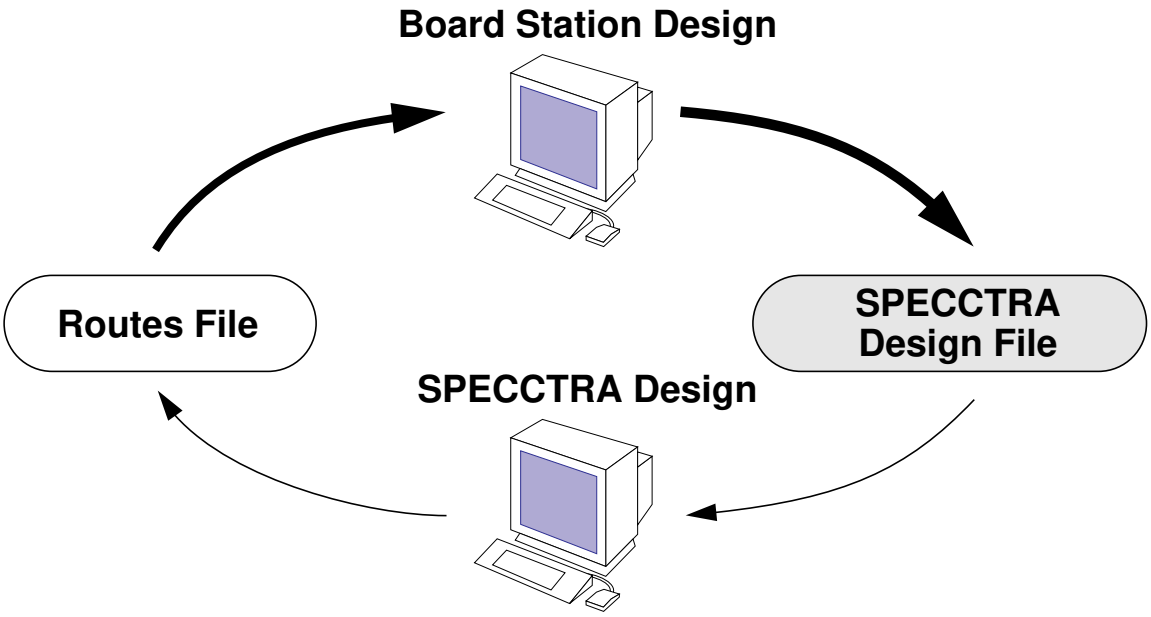


Figure 5-1. Automatic Translation Process from LAYOUT

Figure 5-2 depicts a process flow of shaped-based router files. The default location for storing all of these files is in the *pcb_sb* directory in your design's *pcb* container. The figure focuses on the following four areas in which your interaction with the shape-based router and an awareness of files is important:

- While in LAYOUT, you can create and edit *do* files, using palette icons. You also build the database and generate the SPECCTRA design file, named *art_s_in*. Notice, the shape-based router uses this name as the design name.
- You can invoke the shape-based router with four levels of control. All but the **True Interactive** choice execute a *do* file.
- While in SPECCTRA, you can execute *do* files, use the graphical interface, or enter commands from the keyboard.
- The default names of files created by the shape-based router are shown in parenthesis. The # in the command log file name represents the creation date.

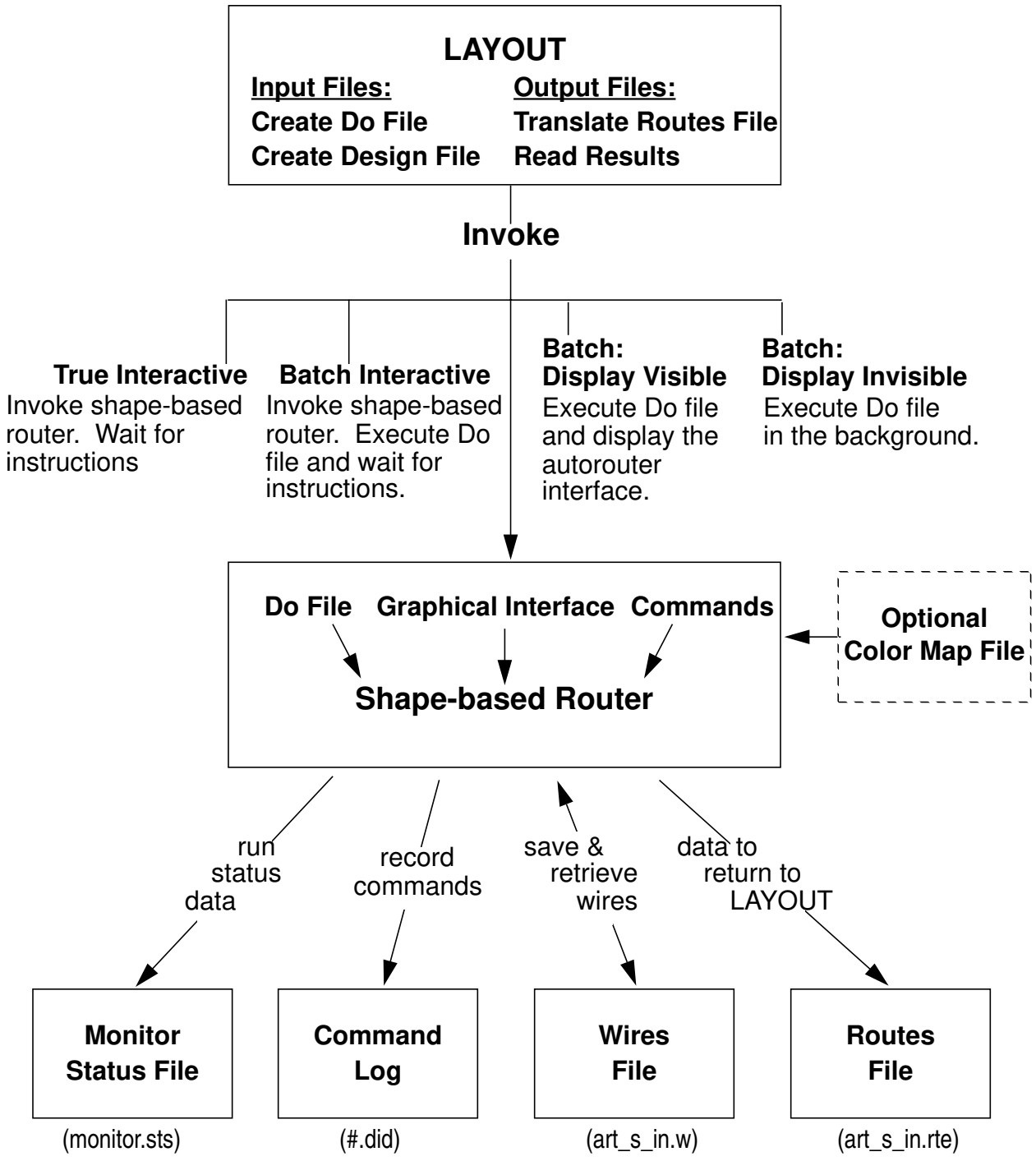


Figure 5-2. Shape-based Router Files

The Graphical User Interface

Figure 5-3 identifies controls and areas of interest in the shape-based router Session window.

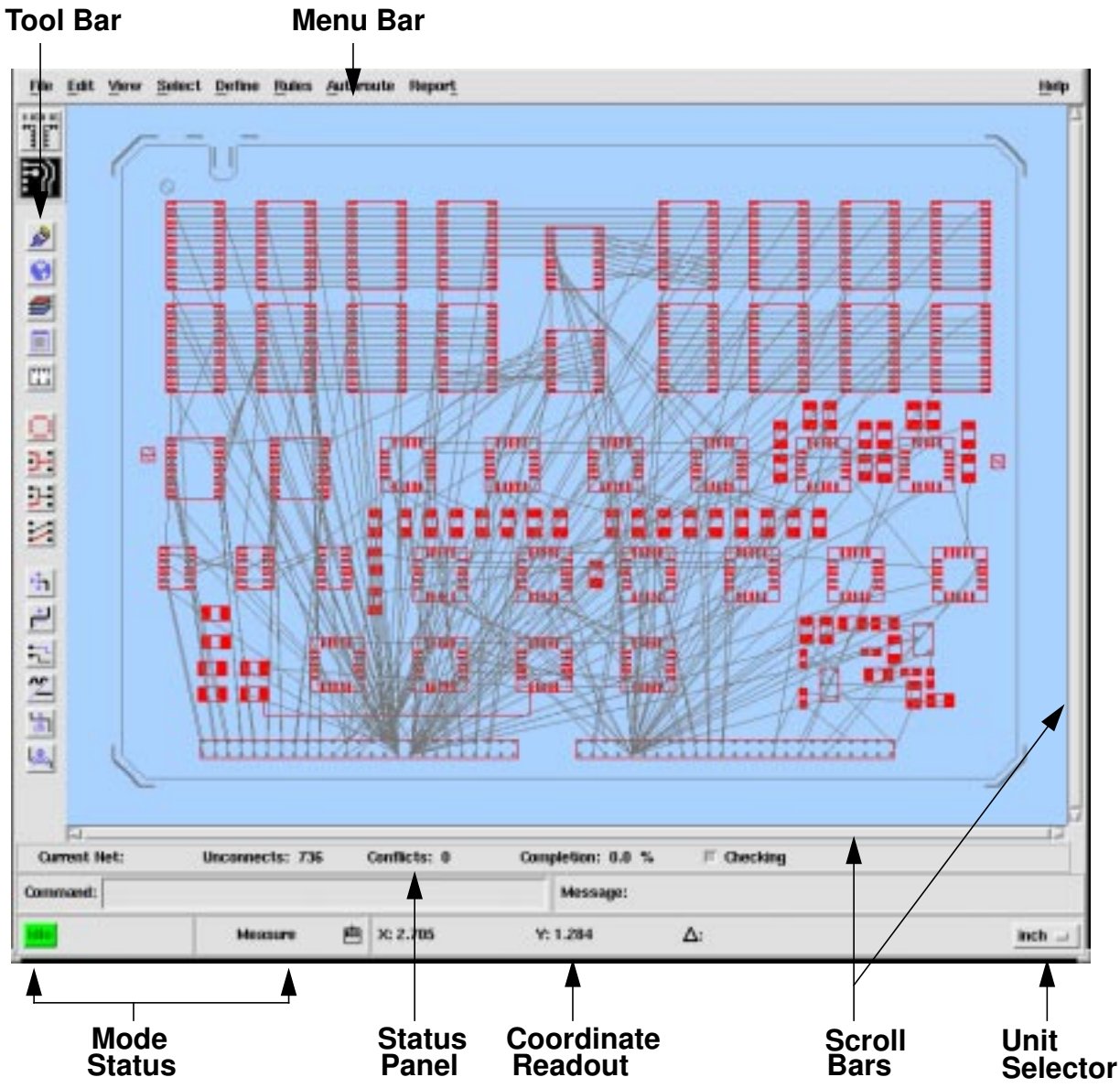


Figure 5-3. Shape-based Router User Interface

Shape-based versus Grid-based Routing

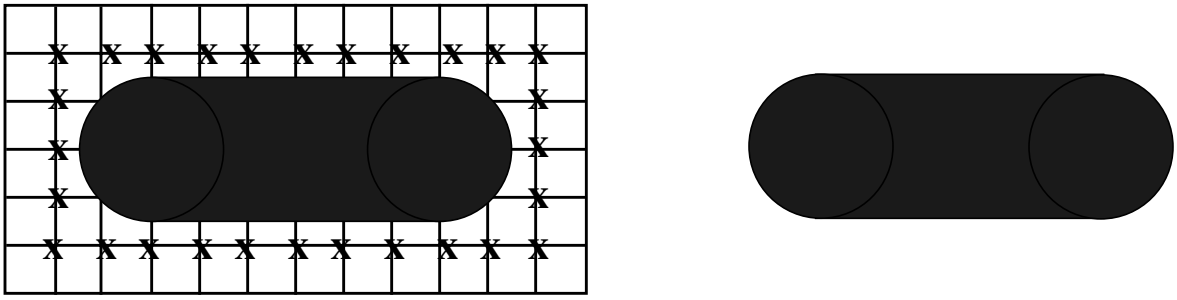
The efficiency and flexibility of shape-based routing often results in high quality routing in less time than grid-based routing.

Comparison to Traditional Grid Map Technology

Design information for shape-based routing is stored in a more efficient, precise form.

- Grid map autorouters decompose information into an array of points, and retain information about each point or grid cell in the array.
- Shape-based autorouters are based on geometric shapes. Design rules and electrical information are attached to each shape.

Figure 5-4 displays the space utilization difference for geometric shapes between grid and shape-based routers.



Approximate dimensions

Exact dimensions:
128 mils x 74 mils

Result: Exact dimensions permit maximum space utilization

Figure 5-4. Shape versus Grid Router Space Utilization

Grid Locations versus Total Shapes

Figure 5-5 displays the difference between a grid map and shapes in terms of the number of items to calculate while routing. A decrease in grid size can dramatically increase the number of grid locations a router must calculate. With shape-based routing, a decrease in grid size represents no change in calculation of area, points, or shapes.

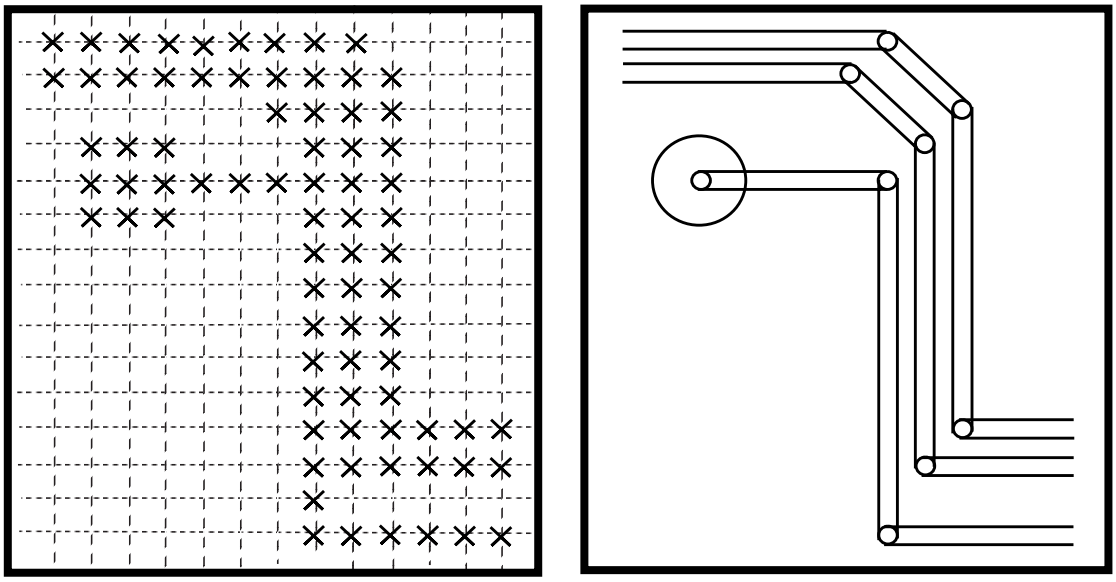


Figure 5-5. Items to Calculate

Consider the two pictures in Figure 5-5. On the left, you see the marked grid locations. On the right, you see the same shapes necessary to replicate the marked grid points. Both use a 20 mil grid with 10 mil traces and 40 mil vias. Contrast the values below shown for grid and shapes, respectively:

195 total grid locations 12 shapes
78 grids marked

Note the dramatic increase when you change the grid to 1 mil:

78000 total grid locations 12 shapes
31200 grids marked

Real Time DRC During Routing

The shape-based router checks shape-to-shape rules during routing. Figure 5-6 displays the shape-to-shape rules in effect during routing, using SPECCTRA vocabulary.

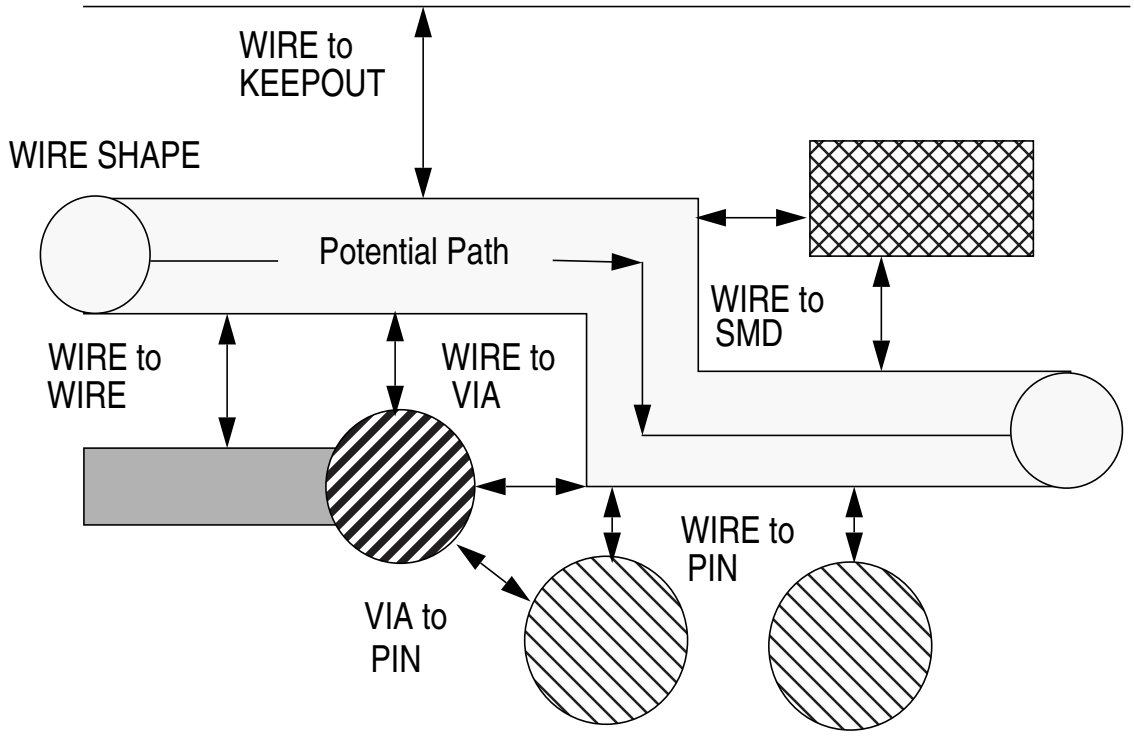


Figure 5-6. Shape-to-shape Routing Rules

Shaperouter Concepts

The shape-based autorouter uses an adaptive routing strategy. That is, it attempts to make all connections on the first pass by allowing conflicts, and then eliminates the conflicts with subsequent passes. The concepts of *conflict* and *routing pass* are important to your understanding of this strategy.

What is a Conflict?

A conflict is the occurrence of two separate signals routed in violation of a clearance rule or intersecting on the same layer.

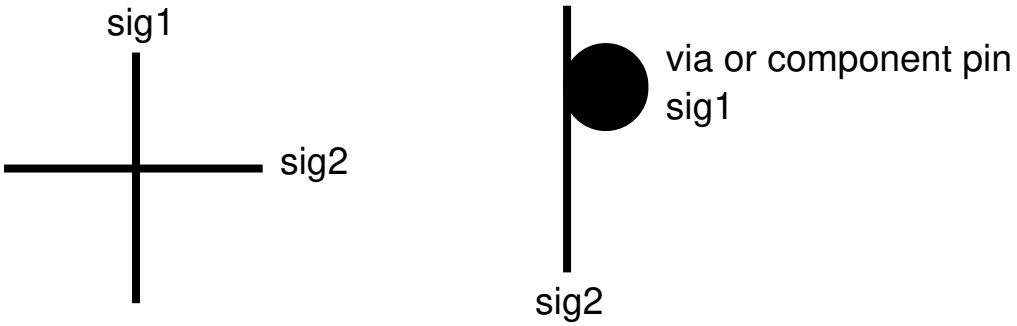


Figure 5-7. Conflict Concept

What are Passes?

A routing pass is one trip through all the *wires to be routed*. For the first pass through the fifth pass, *wires to be routed* includes all connections. For passes subsequent to the fifth pass, *wires to be routed* only includes connections involved in or in the area of conflicts. With each pass, the autorouter gathers information about the problem areas and uses this information to eliminate conflicts.

After each pass, conflicts are marked graphically with a conflict box or a diamond. The number of conflicts after the first pass is typically three to four times the total number of connections. Failures are monitored and recorded in the status file. A failure occurs when the autorouter is unable to find a new path for a connection. During pass one, failures are unroutes. After pass one, failures can consist of unroutes and wires that can't be rerouted with a different path.

Adaptive Routing Strategy

The SPECCTRA adaptive autorouting strategy is a two-phased approach to completing all connections.

The *initial* phase consists of the first five routing passes. The objectives are to create a path for all connections by allowing conflicts and to develop the overall routing flow. The key status file indicators to watch during the initial phase are failures, unroutes, and conflicts.

The *converge* phase consists of the sixth and subsequent routing passes. The goal is to eliminate all conflicts. Only connections involved with conflicts are ripped up and rerouted. If unroutes exist at this state, they probably cannot be completed in subsequent passes. In the converge phase, conflict reduction proceeds at a slower pace. You typically see small percentage reductions of conflicts (less than 30%) during each converge pass. The overall trend is downward for any ten passes.

Figure 5-8 illustrates how the autorouter converges to a solution by eliminating conflicts as the number of routing passes increases. With each pass, an internal costing system exacts a higher price for conflicts. Other routing costs are automatically changed as the autorouter adapts to topology to derive solutions.

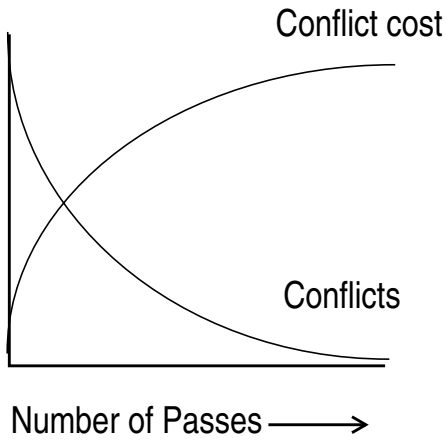


Figure 5-8. An Adaptive Strategy

Shape-based Autorouter Do Files

A *do* file is an ascii file that contains a sequence of shape-based autorouter commands. Three of the invocation choices available for starting the shape-based router in LAYOUT require a do file.

Editing a DO File in LAYOUT

You can create a do file for the shape-based autorouter within the LAYOUT environment. When you edit a do file in LAYOUT, the Palette window displays shape-based autorouter commands. When you click on a command in the Palette window, LAYOUT automatically inserts the command in the do file.

Available shape-based autorouter commands are organized into four lists in the Palette window. The first three lists allow you to create a do file, without worrying about command syntax. If you select a command from one of these lists, a dialog box displays to collect command argument values, and the command is automatically inserted at the cursor location when you execute the dialog box.

When you edit a do file in LAYOUT, the Palette window displays one of the following four lists:

- **Basic Commands**—An alphabetical list of the most commonly used commands.
- **Novice Commands**—A logical ordering of commands that you must include in every do file.
- **All Commands**—An alphabetical list of all commands.
- **Command Syntax**—An alphabetical list of all commands. Selecting a command from this list displays the command syntax without automatically inserting the command.

Before you save a do file in LAYOUT, you can automatically check that the file contains suggested commands.

The Generic Do File

The Palette window provides the option of inserting the following generic do file. It serves both as an example of an appropriate ordering of shape-based autorouter commands and, with the comments, as an explanation of the purpose of some of the commands.

```
# A # in column 1 indicates a comment line. Blank lines are ignored.

# Turn on the automatic save feature and establish the save
# file in the design directory.

bestsave on $/bestsave.sb_w

# Redirect the monitor.sts file

 status_file $/artrouter_status.rpt

# Set global rules -
#  all values hereafter are in inches; set wiring grid to .001"

 unit inch
grid wire .001

# Set temporary rules for initial routing; the via grid is .050"

grid via .050

# Perform initial routing -
#  do bus routing with diagonal corners and fanout SMD power pins

bus diagonal
fanout (pintype power)

# Autoroute 15 passes, set the via grid to .025", then autoroute
# 25 passes with internal costs starting with pass 11.

route 15
grid via .025
route 25 11

 # Perform 2 passes of manufacturing cleanup

 clean 2

# Save wires before recornering and test point assignment

 write wire $/artrouter_wires.sb_w
```

```
# Optionally assign test points and perform a clean pass to consolidate
# (remove the # in column 1 to execute these commands)

#force_via
#clean 1

# Perform diagonal recornering (replace 90 degree corners with135)

recorner diagonal

# Generate an interface (routes) file in the design directory

write routes $/art_s_out.rte

# Save the final status information in a file in the design directory

report status $/artrouter_status.rpt

# Quit the autorouter - do not use this if you want the autorouter
# to remain active

quit
```

Summary

In this module you learned about the shape-based autorouter tool. We compared shape-based autorouting and grid-based autorouting. You learned about shape-based router tool concepts and adaptive strategy. We also discussed the support available in LAYOUT for editing *do* files.

Lab Exercise

In this lab exercise, you explore the process of taking a design from LAYOUT into the shape-based router tool and back into LAYOUT. You explore necessary elements for translation and basic routing commands in the autorouter.

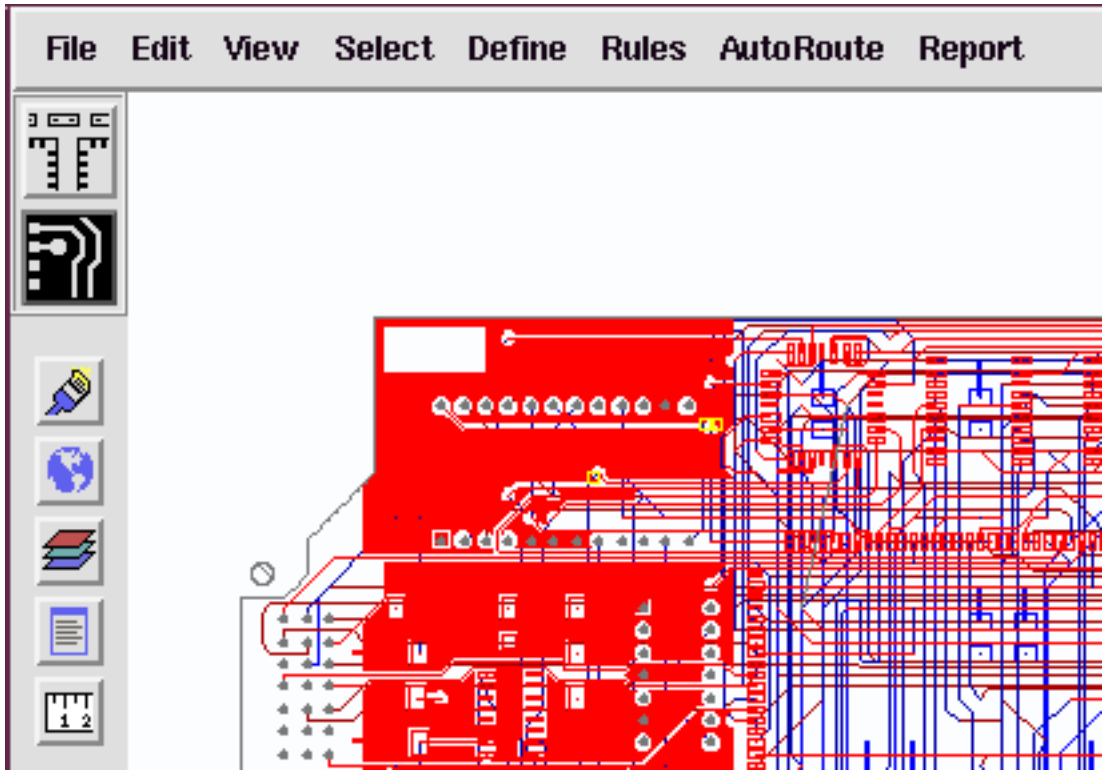
Upon completion of this lab exercise you are able to:

- Invoke the Shape-based Autorouter from LAYOUT.
- Perform basic routing functions.
- Read the Shape-based routing results back into LAYOUT.

Turn to Module 6—Lab 5: "Shape-based Autorouting".

Lab 5
Shape-based Autorouting

In this lab, you use the SPECCTRA shape-based tools to enter basic routing commands and gain familiarity with shape-based autorouting techniques.



Introduction

In this lab exercise, you explore the process of taking a design into the shape-based router and back into LAYOUT. You investigate the necessary elements for translation, basic autorouter commands, and the shape-based router interface.

Upon completion of this lab exercise you are able to:

- Invoke the shape-based autorouter from a LAYOUT session.
- Perform basic routing tasks.
- Execute routing commands with a do file.
- Read the routing data into the LAYOUT session.

Procedure

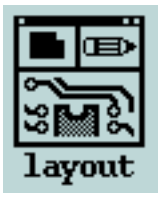
To set up for this lab, invoke LAYOUT and delete all the routing from previous lab exercises.

Preparation for Lab

1. Invoke the Design Manager by entering the following in a shell:

   ```
   $MGC_HOME/bin/dmgr
   ```

2. Navigate to the *board_new* directory by clicking on the four-way icon in the navigator window. In the **Change directory to** dialog box, enter the pathname: your_path/**training/board_new/mod6**. Press the Return key.



3. Invoke LAYOUT on the **sig_az** design and maximize the Session window.

4. Set the Select Filter to ensure you can select **Area Fills**, **Traces**, **Vertices**, and **Vias**.

5. Delete all traces, vias, and fill areas.

Invoking the Shape-based Router

You are now ready to translate the design and invoke the shape-based router using a three step process. In the steps that follow, you complete the following tasks:

- Generate intermediate files used to translate the design.
- Create the SPECCTRA design file.
- Invoke the shape-based router.

1. From the popup menu in the Edit window choose **Top Menu > SPECCTRA > Operations > Collect Design Data**.

   LAYOUT automatically creates the files needed to translate the design into a SPECCTRA design file. This includes unversioned copies of design objects and ascii geometry files.

2. Close the SPECCTRA Gather Files Transcript report window.

3. From the popup menu in the Edit window choose **Top Menu > SPECCTRA > Operations >Build Router Data...**

4. The Build SPECCTRA Database dialog box displays. Set the **Include Traces** button to translate the current net breakup and **OK** the dialog box.

   The Board Station design is translated into a SPECCTRA design file. This is an ascii version of the design information that the shape-based router can read.

5. Look at the contents of the SPECCTRA Database Conversion Transcript and then close the report window.

6. From the popup menu in the Edit window choose **Top Menu > SPECCTRA > Operations >Start Router...**

7. The Start SPECCTRA Auto-Router dialog box displays. Set the **True Interactive** button and **OK** the dialog box.

   The shape-based router Session window appears on top of the LAYOUT Session window. The Transcript window is still visible and can serve as the message area while you use the shape-based router.

   The shape-based router contains a column of icons on the left side of the window. This is referred to as the Tool Bar.

Because control has passed to the shape-based router, you cannot minimize and then maximize the LAYOUT Session window and still use the LAYOUT Transcript window for messages. If you cannot see the Transcript window or if you want to see more lines of the message, you can view the same messages in the message window generated by Design Manager when you invoke LAYOUT.

Autorouting your first board

The purpose of this exercise is to allow you to see the autorouter at work, before you explore its options. You control the shape-based autorouter with commands. The steps that follow direct you to type the commands. You could also use menus and icons or execute a command file, called a *do* file. The command syntax is the same, whether you type the command or include it in a *do* file.

1. Place the cursor in the Command line area, as shown below, and click the Select mouse button..

2. In the Command line area, type:

   route 1

   and press the ENTER key. The autorouter begins routing. Notice that **Pause** appears in the lower-left corner during autorouting.

3. Watch the autorouter as it executes this first pass. The Status Panel tallies the number of attempts and the Edit window displays connections as they are made.

4. At the end of the first pass, notice the yellow conflict markers. A diamond shape indicates a crossover violation and a rectangular shape indicates a clearance violation. Conflicts are resolved during later passes of the autorouter. Expect more conflicts than connections during the first pass.

5. When the **Idle** indicator reappears, in the Command line area, type:

   report status window

   and press the RETURN key.

   A detailed summary of the routing results displays. The file *monitor.sts* automatically records most of this information for you. You can save the detailed summary to a file by replacing *window* with a filename. The Report button in the tool bar also displays this detailed report.

6. Press the **Close** button in the status report window.

Zoom and Pan

Moving about in the shape-based router interface is made easy with Zoom and Pan actions enabled through the middle mouse button.

Zooming in

You can zoom in on an area of interest by pressing and holding the middle mouse button, while moving the cursor up at a diagonal across the area.

1. Locate the cursor below and to left of an area of interest.
2. Press and hold the middle mouse button and drag toward the upper-right corner. A box indicates the selected zoom area. Release the mouse button. The indicated area fills the screen.

   The same action occurs when you drag the cursor from lower-right to upper-left corner.
3. Repeat steps 1 and 2 to zoom in closer.

Zooming out

To zoom out, reverse the motion.

1. Locate the cursor in the upper-left section of the Edit window.
2. Press and hold the middle mouse button and drag toward the lower right. Two boxes appear on the screen. The inside box represents the current zoom area. The outside box increases in size as you drag to the lower-right corner. This box indicates how far you zoom when you release the mouse button.
3. Release the mouse button.

Zooming out to the entire design

There are two methods for redisplaying the entire design. You can click on the World button in the tool bar or use the middle mouse button as described in the following steps.

1. Press and hold the middle mouse button. Two concentric circles appear.
2. Move the cursor horizontally to the left. A single box appears. If two boxes appear, move the cursor vertically until only one box shows.
3. Release the mouse button. The entire design fills the Edit window.

Panning across the work area

You can change the center of view by clicking with the middle mouse button.

1. Move the cursor near the right side of the Edit window. Click the middle mouse button.
2. The point at which you click becomes the new center of the display.
3. Practice panning by clicking the middle mouse button at several locations.

Measure

You use the measure function to measure between any two items in a design and to obtain information about shapes or conflicts.

Measuring between two objects

Ruler Icon

1. Zoom in on two SMD pads so they fill the Edit window.
2. Ensure the mouse is in measure mode. The Mode Status area reads *Measure*. If it does not, Click on the ruler icon in the Tool Bar area.
3. Move the cursor to the edge of one pad.
4. Press and hold the Select mouse button. Move the cursor to the edge of the other pad. Notice the line following the cursor.
5. Release the mouse button. The distance is shown in the message window, as shown in the following example.

```
# Point1 (0.6409, 0.3874) Point2 (0.6409, 0.3624
# Delta=0.025 [INCH] Dx=0 Dy=-0.025
```

Delta = diagonal distance
Dx = x-direction distance
Dy = y-direction distance

The diagonal distance can also be read from the Coordinate Readout Area to the right of the Delta Icon: Δ: diagonal distance

Obtaining information about shapes and conflicts

1. Be sure the mouse is still in measure mode.

2. Click the Select mouse button on any shape, such as an SMD pad or via. The shape information lists in the message window, as shown in the following example:

```
# Point (0.9552, 0.5011) [INCH]
# Rect Xlo=  0.9250 Ylo=  0.4875 Xhi=  0.9750 Yhi=
   0.5125 Layer=s1
# DBU Xlo=2349500 Ylo=1238250 Xhi=2476500 Yhi=1301750
# Comp Pin: U1 4
# Padstack Used: p25x50
```

   If shapes overlap, the data includes definitions of each shape. To get information about conflicts, you must run at least one route pass.

3. Zoom in on a conflict.

4. Move the cursor to the intersection of a crossover conflict or inside the rectangle of a clearance conflict and click.

5. The conflict information is shown in the message window. The readout supplies explicit information regarding the traces or pads involved in the conflict and reports the minimum required clearance.

Routing Connections Interactively

Interactive routing tools offer the flexibility to pre-route connections before starting the autorouter. Also, connections can be interactively edited after the completion of the autorouter.

Route Button

The interactive router can be used in either gridded or gridless mode. It supports online checking of wire width, clearance, and order (daisy-chain) rules.

In this exercise, choose primary and secondary routing layers, choose a routing mode, and interactively route connections. To begin selecting and using route commands, select the route button in the tool bar.

Selecting primary and secondary routing layers

Before you begin interactive routing, you select primary and secondary editing layers. The primary layer is the active signal layer for interactive routing and editing. The secondary layer is the layer the router switches to if you add a via.

A pencil icon is used to indicate primary and secondary routing layers. The secondary layer is indicated by a dimmed pencil icon.

Layer Button

1. Click the Layer button in the tool bar to display the layer panel shown in Figure 5-9.

2. If not already selected, click the layer view buttons for signal layers 1, 2, 3, and 4. A layer is selected when its color shows in the view layer button.

3. Click the pencil icons to see how the dimmed and bold icons toggle to indicate primary and secondary routing layers. Make sure signal layer 1 is the primary routing layer.

   When you begin interactive routing, the primary layer is the active layer on which you are routing or editing.

4. Check your layer panel display against the figure and then **Close** the layer panel display.

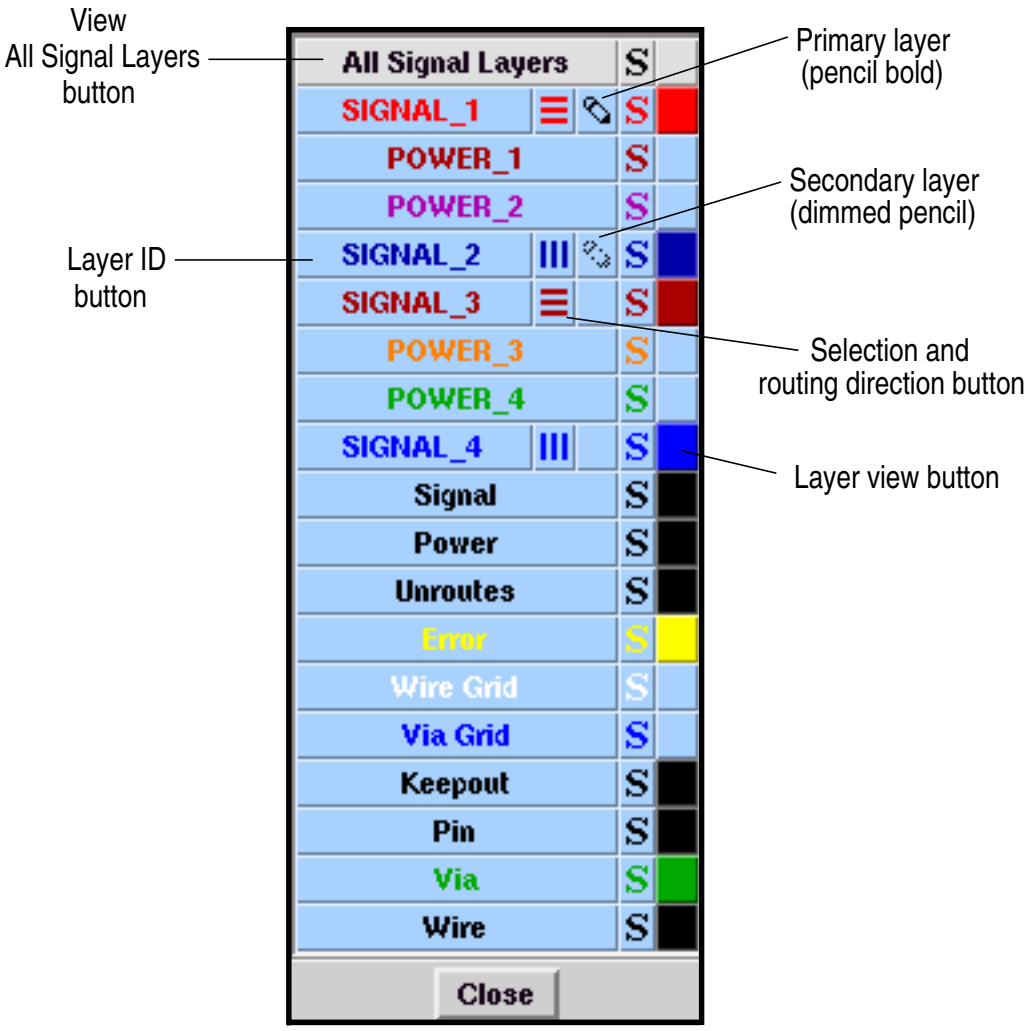


Figure 5-9. Layer Panel Example

Choosing an Interactive Routing Mode

The Select mouse button performs different functions depending upon the interactive routing mode you choose. When you press the right mouse button in the Edit window, the Interactive Route popup menu appears.

The following table describes the modes.

Edit Route	Creates a new wire path and adds vias. Movable wires are shoved aside to add the new path if Push is checked in Setup.
Move Route	Moves wires or vias. All movable wires in the path of the wire or via you are moving are shoved aside if Push is checked in Setup.
Copy Route	Copies an existing wire to an unroute with a similar length and path.
Critic Route	Removes extra bend points in a single wire or in several wires if you draw a bounding box.
Cut Segment	Break a single segment in two.
Delete >	Segment Mode—removes a single wire segment. Wire Mode—removes a complete wire (all segments) and attached wires. Net Mode—removes all wires and vias of a net. Repair Net Mode—removes all wires on a net that violate fromto order rules.

Each mode can also be selected from the tool bar:

Edit Route

Move Route

Copy Route

Critic Route

Cut segment

Delete

Routing Wires

You can route new wires, and add, change, or replace existing wires. When you are in Edit Route mode, an envelope surrounds the wire segment you are digitizing and indicates the clearance rule in effect. Arrows and alignment marks are also added to the display to help you line up the cursor with the nearest wire, pin, or via on the net.

1. Choose Edit Route Mode from the tool bar or the right button menu.

2. Click on a pin, a via, a guide, or a wire to start.

   If you start editing from a via or a through-pin, you can choose the primary layer. If you start from an SMD pad or a wire, the primary layer is automatically set by the router.

3. Move the pointer to the next location and click to add the wire segment.

While you are routing a connection, a preview wire stretches from the last point digitized as you move the pointer. The preview wire automatically jogs around shapes that are not part of the same net and shows the intended path before you actually digitize a segment.

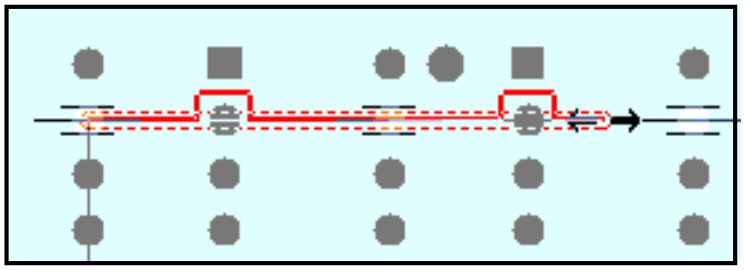


Figure 5-10. Edit Route Preview Wire

Before you complete a connection, you can delete a portion of the segment you are routing by tracing back over the part you want to remove and clicking. The segment is trimmed to the cursor position.

You can interrupt your routing and leave the segments already created, or remove the segments.

Start by routing a wire. Click on a pin, a via, a guide, or a wire to start.

After you have created one or more segments, press the right mouse button. As shown in Figure 5-11, the Edit Route popup menu appears.

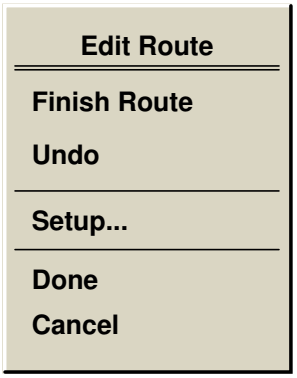


Figure 5-11. Edit Route Menu

To interrupt your routing and retain an unfinished wire

- Press the right mouse button and select **Done** from the Edit Route menu.

To interrupt and remove unfinished wiring.

- Press the right mouse button and select **Cancel** from the Edit Route menu. You can achieve the same result by selecting another mode from the tool bar.

Moving a Wire

Move Route Button

Use Move Route to move a segment or via orthogonally, or to move a corner of a wire diagonally. Segments connected to whatever is attached to the pointer stretch and adjust accordingly. Other wires in the path of a moving wire or via are pushed aside if the Push check box is checked in Setup.

1. Choose Move Route Mode from the Edit Route menu or from the tool bar.
2. Click on the wire segment or via you want to move.

   The wire or via object attaches to the cursor. As you move the pointer the object moves, connected wires stretch to follow the cursor, and other wires in the path are pushed aside to maintain required clearances.
3. Click the Select mouse button to place the via or wires at the location under the cursor.

Try moving a few wires and vias before moving on to the next task.

Creating a New Segment in a Wire

The Cut Segment Mode is used to create a new segment in a wire. Usually, you create a new segment so you can move just a portion of a wire. You must cut a segment at two locations to break it into three pieces, thereby defining the segment piece to move. The location of each cut is marked with an X until you repaint.

To create a new segment in a wire:

1. Choose Cut Segment Mode from the Interactive Route Menu.
2. Click at two locations on a wire segment.
3. Select the Move Route Mode and move the cut segment.

Figure 5-12 shows a Cut Segment being used in a Move operation

.

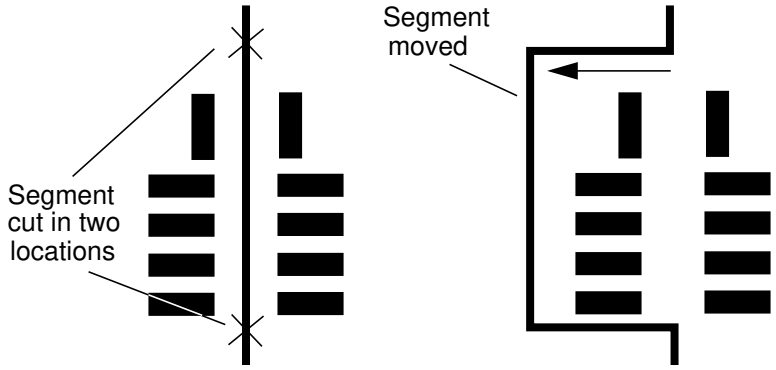


Figure 5-12. Example of Cut Segment

Executing Commands with a Do File

All of the commands executed in the autorouting exercise can be combined, along with other control commands, in a *do* file. You can create and edit a do file in LAYOUT, and execute the file from the Start Shaperouter Auto-Router dialog box.

The Start Shaperouter Auto-Router dialog box in LAYOUT includes three choices for immediately executing a do file when invoking the autorouter. If you choose **Batch Interactive**, the autorouter remains active after it executes all commands in the do file. Be sure not to include the Quit command in a do file, if you want to view the results in the shape-based router.

In this exercise, you execute the do file shown below.

```
bestsave on $/best.w
status_file $/route.sts
unit mil
grid smart (wire 1) (via 1)
bus diagonal
route 25
clean 2
route 50 16
clean 4
write wire $/sig_az.wir
write routes $/sig_az.rte
report status $/sig_az.sts
```

1. From the Menu Bar at the top of the autorouter window, select **File > Execute Do File...** The **Execute Do File** dialog box displays.

2. The dialog box defaults to the correct location. Change the file name to *sig_az.do* and execute the dialog box.

   Watch the message window and you can see the do file commands execute. Because it takes more time than is available to execute the entire do file, the next step tells you how to interrupt the autorouter.

3. Press the **Pause** indicator. The indicator changes to show **Continue** and **Stop**. Press the **Stop** button to end the routing.

Returning to LAYOUT

Before returning to LAYOUT, you must save the routing results as a SPECCTRA routes file.

1. Select **File > Write > Routes** and execute the Write Routes dialog box to save the routing results in the default location.

2. Select **File > Quit**. Press the **Quit** button in the Confirm Quit dialog box or Press the RETURN key.

   The shape-based router window closes. If you have not minimized LAYOUT, the Shaperouter Session Transcript report displays.

3. Close the Shaperouter Session Transcript report window.

4. From the popup menu in the Edit window choose **Top Menu > Shape-Based Router > Operations > Convert Trace Data...**

5. The Convert Shape-based Routes Object dialog box displays. Because it defaults to the proper location, execute the dialog box

   LAYOUT translates the information in the routes file, sorts it into design object categories, and creates the files used to restore the shape-based traces.

6. Close the Database Conversion Transcript report window.

7. From the popup menu in the Edit window choose **Top Menu > Shape-Based Router > Operations > Load Routed Traces...**

8. The Restore Shape-based Routed Traces dialog box displays. Set the **Add to Existing Traces** button and **OK** the dialog box.

   LAYOUT restores the shape-based generated traces.

9. Exit from LAYOUT. You can choose to save the translation results in design objects by saving the design, or exit without saving.

You now return to the Design Manager. The next module is *Creating Manufacturing Data*.

Appendix A: Autorouter Costs and Schedules

Cost settings control how traces are routed. They determine whether vias, bends, and other elements go into each connection. The autorouter scans the grids from source to target, looking for the least costly route. Scanning involves pausing at each grid point, calculating cost to that point, and seeing where to move next. When the least costly route is found, the connection is routed. The autorouter then attempts the next connection.

As the autorouter scans, a running total is dept. Updates are made at each grid point keeping the total cost the autorouter spent getting to that grid from the source. The total is made up of individual cost of vias, bends, and other cost items used. If none of the seven cost items has been used, the total is zero.

Each connection has a budget of 1000 cost units. That means total cost for that route must not exceed 1000. Costs add up quickly. If you sketch some typical routes and add the costs, you might find that if your individual costs are too high, 1000 cost units are spent quickly and routes fail.

After scanning all potential paths if the autorouter determines that a connection cannot be routed within budget, the connection fails and becomes unrouted. It is not tried again until the next pass.

Cost Values

Each cost schedule contains a value for each of the following costs.

- **Way**—cost of moving the wrong way on a routing layer. For example, moving vertically on a horizontal layer.
- **Excess**—cost of moving away from the target.
- **Diagonal**—cost of using a diagonal segment.
- **Bend**—cost of making a bend in a trace.
- **Via**—cost of using a via to continue a route on another layer.
- **Intersection**—cost of crossing traces during rip-up routing.
- **Near**—cost of clearance violations during squeeze-through routing.
- **Site**—cost of routing a segment over a via location.
- **Fence**—an additional parameter limiting the area the autorouter uses to complete each connection. Fence defines the bounds within which the autorouter must work.

You can define your own schedule of values for each cost with the $$template_schedule() builtin function.

The autorouter is EXTREMELY sensitive to this cost setting. Careless settings can drastically affect completion rates and autorouter execution times.

How Costs are Totaled

The autorouter considers its choices at each grid point. If diagonal traces are not allowed, the autorouter can choose from four moves. If diagonals are allowed, the autorouter can select from eight moves.

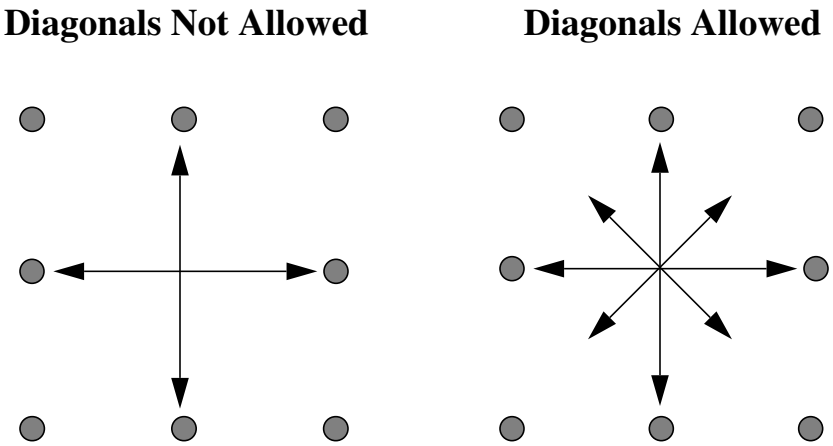


Figure A-1. Possible Router Moves

The autorouter keeps a running total. The total updates at each grid point. When the autorouter finds the least costly route, the autorouter routes the trace. If the cost of a candidate path exceed 1000, that path fails. If the autorouter cannot find any path whose cost is less than 1000, the route fails.

Orthogonal segments (vertical and horizontal) have no cost, provided they are not wrong way traces, do not interfere with potential via sites, or are not going away from the target. The moves shown in Figure A-2 are cost free.

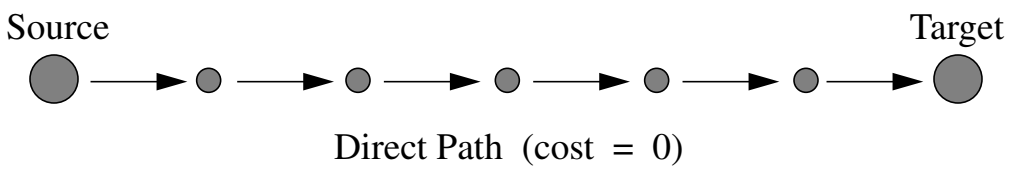
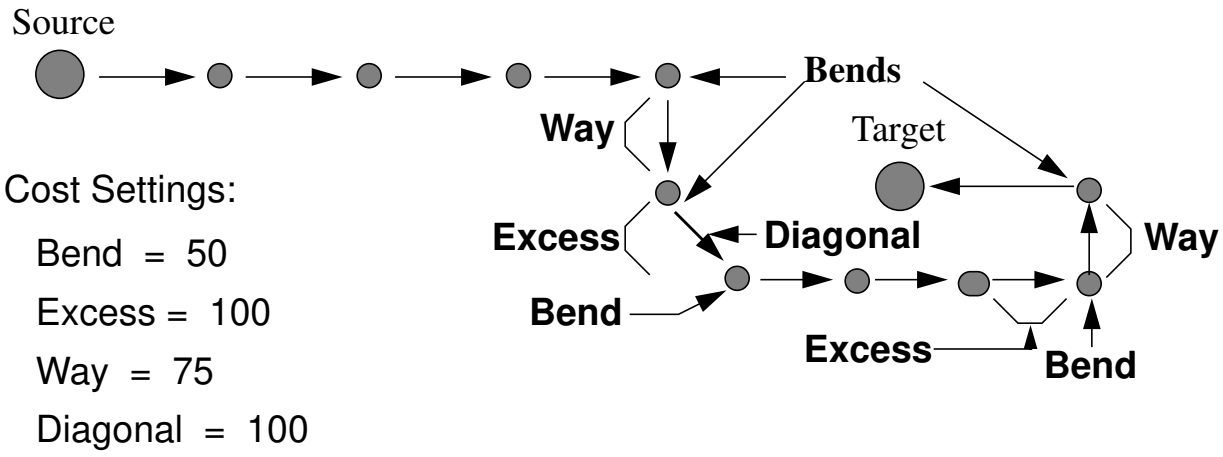


Figure A-2. Cost-free Moves

Bends, diagonals, wrong way, and excess routes do cost. The moves shown in Figure A-3 would result in a total of 700 cost units.



Total Cost = (5 X 50) + (2 X 100) + (2 X 75) + (100) = 700

Figure A-3. Moves that Cost

Routing Cost Schedules

The autorouter selects a particular schedule of costs depending on which router (pattern, automatic/rip-up, or manufacturing) you use, and the percentage of the board that is routed.

Pattern Router:

The pattern router has eight schedules. The pattern router routes based on each schedule in the sequence shown, unless you specifically choose a particular cost schedule.

Table A-1. Pattern Router Cost Schedules

Schedule	Inter	Way	Diag	Via	Site	Excess	Bend	Fence
0	1000	50	40	1000	0	2	2	1
1	1000	50	50	1000	0	2	5	1
2	1000	50	50	1000	0	2	5	2
3	1000	200	200	400	0	2	5	3
4	1000	100	100	200	0	2	5	4
5	1000	50	50	100	0	2	5	5
6	1000	20	20	40	0	2	5	8
7	1000	10	10	20	0	2	5	10

Automatic Router:

The automatic, or rip-up and retry, router has 15 schedules. Before each pass, the autorouter determines the percentage of completed routing and selects a schedule based on that percentage. For example, if no board connections are complete, the autorouter uses the first schedule of default costs. The next time you invoke the autorouter, it determines the percentage of completed routes. If the board is 75% complete, the autorouter uses schedule #3 of default costs; if the board is 87.5% complete, the autorouter uses schedule #7.

Table A-2. Pattern (Rip-up and Retry) Router Cost Schedules

Schedule	Inter	Way	Diag	Via	Site	Excess	Bend	Fence
1	10	3	3	25	0	2	3	1
2	15	2	2	24	0	1	3	2
3	20	1	1	23	0	1	3	3
4	35	0	0	22	0	1	3	4
5	50	0	0	21	0	1	3	5
6	80	0	0	20	0	1	3	8
7	120	0	0	19	0	1	3	10
8	130	0	0	18	0	1	3	10
9	150	0	0	18	0	1	3	10
10	180	0	0	18	0	1	3	12
11	200	0	0	18	0	1	3	12
12	300	0	0	15	0	1	3	14
13	400	0	0	15	0	1	3	14
14	500	0	0	15	0	1	3	15
15	1000	1	1	15	0	1	3	2

Table A-3. Squeeze Through and Shove Aside Cost Schedule

Schedule	Inter	Way	Diag	Via	Site	Excess	Bend	Fence
16	1000	0	0	50	0	1	5	2

Manufacturing Router:

The manufacturing router has only one schedule.

Table A-4. Manufacturing Cost Schedule

Schedule	Inter	Way	Diag	Via	Site	Excess	Bend	Fence
17	1000	1	0	50	0	1	5	2

INDEX

INDEX [continued]

S

U

V

欢 迎 光 临

http://www.mweda.com　　http://edastudy.ik8.com

本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

破解软件类：微波仿真软件

ADS2004A　ADS2003C　ADS2003A
HFSS9.2　HFSS9.1　HFss9.0　HFSS8.0
Ansoft Designer1.1　Ansoft Serenade8.71　Ansoft Maxwell 10　Ansoft SIWave
Microwave Office2002　Sonnet Suite Pro 9.52　CST5.0
Super NEC 2.5　Zeland IE3D9.2　XFDTD 6.0

破解软件类：PCB 工具软件

Mentor EN2004　Mentor EN2002　Mentor ePD2004　Mentor SDD2004
Mentor WG2004　Mentor WG2002　Mentor ISD2004
PowerPCB5.0　PowerLogic5.0　Orcad10.3
PADS2005　PADS2004
Cadence SPB15.2 (Allegro 15.2)　Cadence PSD15.0 (Allegro 15.0)

软件学习、培训教程：

Mentor EN: Mentor EN 原版培训教程
Mentor EN 视频教程

Mentor WG: Mentor Expedition / Mentor WG 中文用户手册
Mentor Expedition (WG) PCB Training Workbook
Mentor DxDesigner Design Processing Training Workbook

HFSS： HFSS 9.2 入门与提高教程
HFSS 9.0 入门与提高教程
HFSS9 视频教程
HFSS9 Training Tutorials
HFSS8 Training Manual
电子科大 HFSS8 教程

ADS: ADS 中文基础教程
ADS 设计实验教程
ADS 入门教程
ADS2003 Training Workbook: ADS2003 fundamental
ADS Training Workbook for Momentum
ADS Customization Training Workbook

Using ADS Communication Systems Designer

Using ADS to Design WCDMA/3GPP Communication Systems

Ansoft Designer:

Ansoft Designer 2.1 Full Book

Ansoft Designer System Training Guide

Ansoft Designer 1.1 入门与提高

Ansoft Designer 多媒体教程

Ansoft Designer Training 讲义

Cadence Allegro:

Allegro 15.2 原版教程

Concept-HDL 15.2 原版教程

Allegro 14.2 原版教程

Allegro 视频教程

Allegro PCB Layout 高速电路板设计

注： 本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D，Zeland IE3D, PowerPCB，Orcad等的培训教程。以及射频/微波/天线类英文原版书籍。详情可登陆：http://www.mweda.com 或 http://edastudy.ik8.com